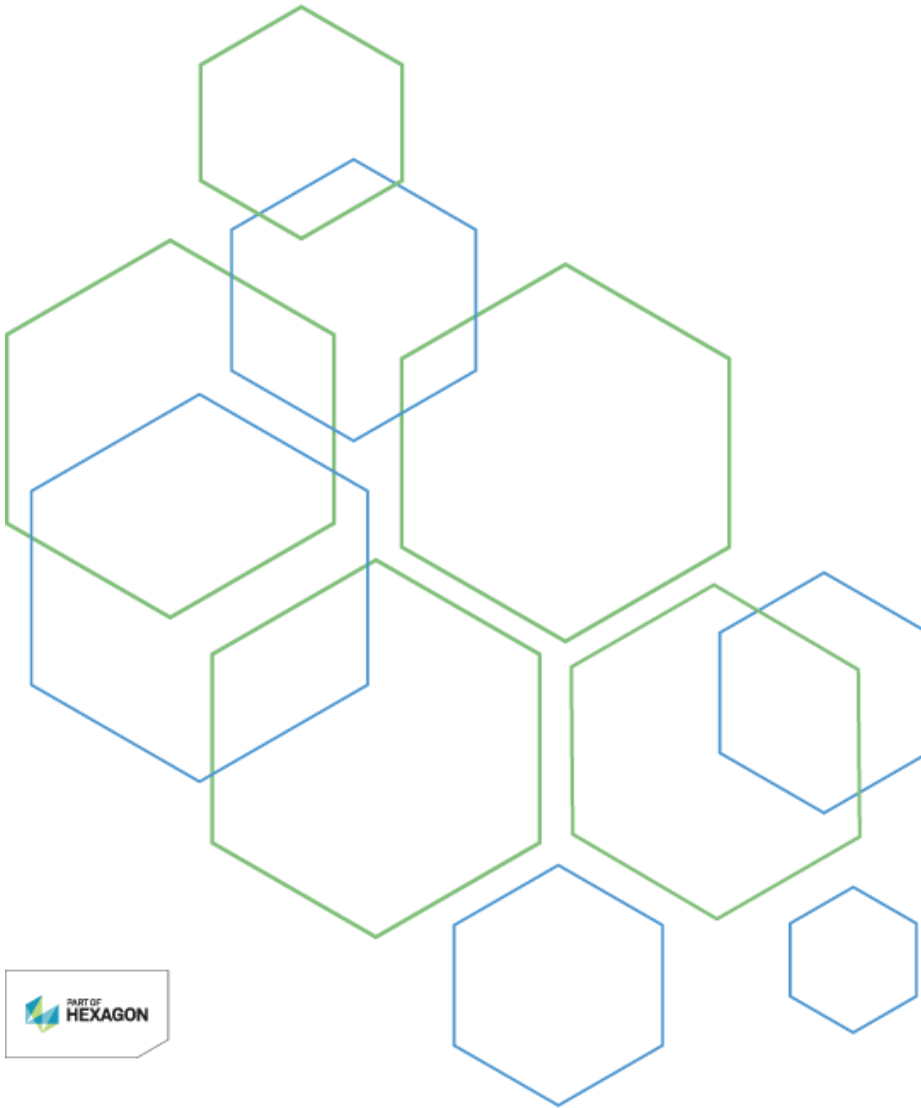


INTERGRAPH®
Smart ➞ **3D**
Structural Manufacturing
User's Guide



Version 2016 (11.0)
November 2016

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Preface

This document is a user's guide for the Structural Manufacturing functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.


Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (<https://smartsupport.intergraph.com>).

What's New in Structural Manufacturing

The following changes have been made to the Structural Manufacturing task.

Version 2016 (11.0)

- The **Delete Optional** property in **Model Data Transform** now supports structural manufacturing entities. (P1 CP:256797)
- Added three new properties, **Nearest Corner Point**, **Horizontal Distance to Nearest Corner Point**, and **Vertical Distance to Nearest Corner Point**, to the **Pin Jig 3D View** dialog box. For more information, see *Pin Jig 3D View Dialog Box* (on page 189). (P2 CP:266128)
- You can now group structural manufacturing objects with WBS. (P2 CP:268825)
- Added the **Submit Batch Job** command to run batch processes using the Intergraph Batch Services framework. For more information, see *Submit Batch Job* (on page 335). (P2 CP:273093, P2 CP:178040)
- Added a new locate filter, **Construction Graphics**. For more information, see *Selecting Objects* (on page 19). (P2 CP:271166)
- A new template .xml output, SMS_PLATE, supports annotations. (P2 CP:272575)
- The Annotation Editor now supports panel-line objects. For more information, see *Annotation Editor* (on page 306). (P2 CP:277667)
- Added two new options, **Generate Unfold Data** and **Frame System** to the **Copy Structure Ribbon**. For more information, see *Copy Structure* (on page 354). (P3 CP:291554)
- The **New (Block) Assembly** command is now available in the Structural Manufacturing task. For more information, see *New (Block) Assembly* (on page 322). (P3 CP:278504)
- Added the **Managed Export** commands to generate and manage XML and DSTV output for parts. For more information see *Managed Export for DSTV* (on page 342), *Managed Export for Smart Production* (on page 342), *Managed Export for SMS_SCHEMA*, and *Managed Export* (on page 343). (P3 CP:247834)
- The **Check Manufacturability** command is now available in the Structural Manufacturing task. For more information, see *Check Manufacturability* (on page 359). (P3 CP:290951)
- The **Macro Editor** command is now available in the Structural Manufacturing task. For more information, see *Macro Editor* (on page 352). (P3 CP:290751)
- A new shrinkage type **By Vector** is now available. In addition, the **Global Axis** shrinkage type has been renamed **By Axis**. For more information, see *Shrinkage* (on page 129). (P3 CP:268521)
- Updated the descriptions of the pin jig properties. For more information, see *General Tab (Pin Jig Properties Dialog Box)* (on page 195). (P3 CP:278351)
- Added information on the **Pick Edges**  option for selecting boundaries. For more information, see *Boundary Methods* (on page 43). (P3 CP:289423)
- Added a new option, **MfgPlate/MfgProfile unfolding semantic log**, to the **Structural Manufacturing Error Log Enable** dialog box. For more information, see *Unfolding Tab (see "Unfolding Tab (Structural Manufacturing Error Log Enable Dialog Box)" on page 371)*. (P3 CP:292030)

- Added the **Manufacturing Panel** command to support panel-line production. For more information, see *Manufacturing Panel* (on page 218). (P4 CP:248100)

SECTION 1

Structural Manufacturing

The Structural Manufacturing task creates manufacturing data from the model. This data is used to create and construct the physical parts of the model. This is accomplished by adding manufacturing-specific information to the detailed parts. After that, the manufacturing parts and supporting parts are generated by rules.

The manufacturing-specific information includes:

- Fabrication Margins
- Assembly Margins
- Shrinkage
- Manual marking lines (The majority of marking lines are generated automatically by rules.)
- Manufacturing Tabs

The manufacturing supporting parts include:

- Templates
- Pin jigs














Manufacturing data is also controlled by applying rule-based processes and rule-based automated marking on manufacturing parts, such as:











- Tabs and bridges
- Plate bevels
- Unfolding algorithms for curved plates
- Roll lines
- Location marks for attached structure
- Reference plane marks

Manufacturing data is output in several ways:

- XML, which can be read by external software such as nesting software.
- Profile sketches for profile parts, created in the Drawings and Reports task. For more information, see the *Drawings and Reports User's Guide*.
- Template set drawings and reports, created in the Drawings and Reports task. For more information, see the *Drawings and Reports User's Guide*.
- Pin jig drawings and reports, created in the Drawings and Reports task. For more information, see the *Drawings and Reports User's Guide*.

You can start the Structural Manufacturing task by clicking **Tasks > Structural Manufacturing**. The Structural Manufacturing task has these commands:


	Select - Used to select objects in the model. For more information, see <i>Selecting Objects</i> (on page 19).
	Marking - Adds manual marking lines to detailed parts, such as Block Control, Painting Line, and Robot. For more information, see <i>Marking</i> (on page 65).
	Marking by Definition - Creates marking lines based on symbol definition. For more information, see <i>Marking by Definition</i> (on page 89).
	Fabrication Margin - Adds fabrication margin to detailed parts at plate edges or profile ends. You can place constant value margins as well as oblique margins. These margins extend along the full edge of the plate or at the end of a profile that you select. For more information, see <i>Fabrication Margin</i> (on page 93).
	Connect Fabrication Margin - Selects all of the plates or profiles bounded to the plate system or part, and selects all the faces at the bounding plate systems or parts to place margins. For more information, see <i>Connect Fabrication Margin</i> (on page 105).
	Margin by Assembly Connections - Selects all of the plates or profiles bounded to the plate system or part, or selects all of the plates or profiles bounded to an assembly, and then selects all the faces at the bounding plate systems or parts to place margins. For more information, see <i>Margin By Assembly Connections</i> (on page 115).
	Regional Margin - Adds a margin over a region of an edge. You can create multiple regional margins along a single edge. For more information, see <i>Regional Margin</i> (on page 122).
	Shrinkage - Adds extra material to detailed parts to account for weld shrinkage by scaling either along the edge or by axis. For more information, see <i>Shrinkage</i> (on page 129).
	Multi Shrinkage - Stretches multiple manufacturing parts to account for steel shrinkage during welding. For more information, see <i>Multi Shrinkage</i> (on page 137).
	Manufacturing Tab - Adds manufacturing tabs to detailed parts at plate edges. You can place tabs of various types. For more information, see <i>Manufacturing Tab</i> (on page 142).
	Plate and Profile Face Template - Creates a set of templates for detailed plate and profile parts at positions that you specify. The command generates the template shapes and template table data. For more information, see <i>Plate Template</i> (on page 148) and <i>Profile Face Template</i> (on page 162).
	Tubular Member Template - Creates and modifies templates to check the curvature of tube member end cuts. For more information, see <i>Tubular Member Template</i> (on page 168).
	Edge Template - Creates and modifies templates to check the curvature of edges. For more information, see <i>Edge Template</i> (on page 177).

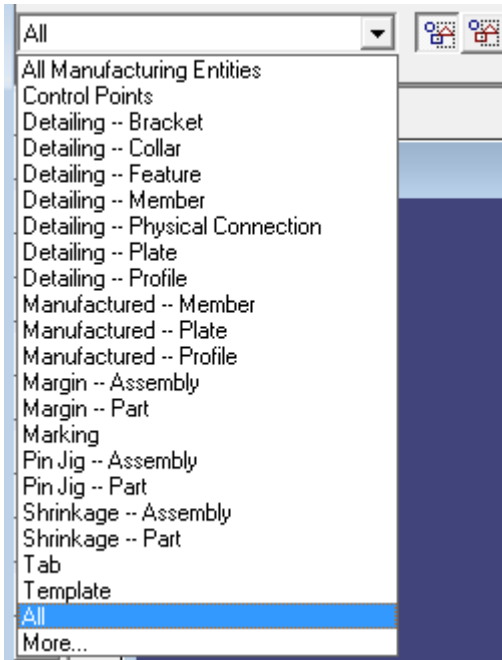
	Pin Jig - Creates a set of pin jigs for detailed plate parts or assemblies at defined positions that you specify. For more information, see <i>Pin Jig</i> (on page 181).
	Manufacturing Plate - Creates a two-dimensional wireframe representation of a detailed plate that has manual and automated rule-based manufacturing data such as margins, shrinkage, marking lines, beveling, and so on applied. For more information, see <i>Manufacturing Plate</i> (on page 204).
	Manufacturing Panel - Creates a two-dimensional wireframe representation for detailed plate parts combined into a panel to support panel-line production. For more information, see <i>Manufacturing Panel</i> (on page 218).
	Manufacturing Profile - Creates a two-dimensional wireframe representation of a detailed profile that has manual and automated rule-based manufacturing data such as margins, shrinkage, marking lines, beveling, and so on applied. For more information, see <i>Manufacturing Profile</i> (on page 234).
	Manufacturing Member - Creates a two-dimensional wireframe representation of a detailed member that has manual and automated rule-based manufacturing data such as margins, shrinkage, markings lines, beveling, and so on, applied. For more information, see <i>Manufacturing Member</i> (on page 251).
	Part Monitor - Displays the manufacturing part as XML or as graphics interpreted from XML, in order to preview manufacturing data before it is output. The XML is created as the command is executed, but is not saved in the database. For more information, see <i>Part Monitor</i> (on page 272).
	Annotation Editor - Moves annotations to avoid overlap, deletes unnecessary annotation, and adds annotations for clarity during production. For more information, see <i>Annotation Editor</i> (on page 306).
	Manufacturing Service Manager - Updates or reviews objects. For more information, see <i>Manufacturing Service Manager</i> (on page 317).
	New (Block) Assembly - Designs assemblies and assembly blocks and defines properties for their associated building method. For more information, see <i>New (Block) Assembly</i> (on page 322).
	Batch Service Manager - Schedules a service manager batch job. This command is available on the right-click menu in the Workspace Explorer for Blocks and Assemblies . For more information, see <i>Batch Service Manager</i> (on page 266).
	Part Editor - Edits manufactured parts. You can add markings, contours, and annotations to manufactured parts, and then save those changes to the database. For more information, see <i>Part Editor</i> (on page 291).
	MFG Environment Display - Controls what structural manufacturing aspects are displayed in the graphic view. This command is available on the Tools menu. For more information, see <i>MFG Environment Display</i> (on page 271).

	Submit Batch Job - Runs batch processes using the Intergraph Batch Services framework. This command is available on the Tools menu. For more information, see <i>Submit Batch Job</i> (on page 335).
	Export to Nesting - Translates the geometry of selected manufacturing data exchange formats and stores this information in a specified file. The manufacturing data is not saved in the Model database. This command is located on the Tools > Manufacturing Export menu. For more information, see <i>Export to Nesting</i> (on page 340).
	Managed Export for DSTV - Exports the DSTV data for the selected assembly, and manages the .nc1 files that the software generates. This export uses the DSTVConfig.xml default configuration file. This command is located on the Tools > Manufacturing Export menu. For more information, see <i>Managed Export for DSTV</i> (on page 342).
	Managed Export for Smart Production - Exports both the DSTV and XML data. This export uses the SmartProductionConfig.xml default configuration file. This command is located on the Tools > Manufacturing Export menu. For more information, see <i>Managed Export for Smart Production</i> (on page 342).
	Managed Export for SMS_SCHEMA - Exports the SMS_SCHEMA .XML data for the manufactured parts of the selected assembly. This export uses the SMS_SCHEMAConfig.xml default configuration file. This command is located on the Tools > Manufacturing Export menu. For more information, see <i>Managed Export for SMS_SCHEMA</i> .
	Managed Export - Creates a neutral export. This export does not have an associated configuration file. Instead, it uses the configuration file that you select. This export is intended to be used with custom configuration files. This command is located on the Tools > Manufacturing Export menu. For more information, see <i>Managed Export</i> (on page 343).
	Custom Processing - Runs custom developed (Visual Basic) reporting software. This command is available on the Tools menu. For more information, see <i>Manufacturing Services</i> (on page 344).
	Navigator - Provides a viewer to look at the individual graphic elements that are part of pin jigs and templates. This command is available on the Tools > Struct Manufacturing menu. For more information, see <i>Navigator</i> .
	Stiffeners Diagonal Measurement - Provides a tool to extract distance information between two stiffeners or a pin jig for accuracy verification. This tool also provides information on mounting angles. This command is located on the Tools > Struct Manufacturing menu. For more information, see <i>Stiffeners Diagonal Measurement</i> (on page 346).

	<p>Select Connected Object - Provides a tool to check which objects are connected to a selected object based on physical connections between the parts. This allows you to check if a given manufacturing plate or profile is out of date. This command displays the same list as that shown on the Status tab when you modify a single manufacturing plate or profile. This command is located on the Tools > Struct Manufacturing menu. For more information, see <i>Select Connected Object</i> (on page 350).</p>
	<p>Copy Structure - Provides a tool to generate new plate geometry by copying from an original plate with advanced plate system marks representing the connected structure. This allows you to product the manufacturing data bypassing any problems from the original model data set. Also provides a tool to generate a surface from plates that can be taken to GAR and repaired before it is used for manufacturing data. This command is located on the Tools > Struct Manufacturing menu. For more information, see <i>Copy Structure</i> (on page 354).</p>
	<p>Macro Command - Provides a tool to view, edit, and overwrite generated macros placed on profile and member parts. The software displays profile and member parts and the macros generated for the parts in a grid. You can edit macro attributes in the property grid, or apply a different macro to a part. You can also reset the definition and attributes of a macro to the rule-selected values. This command is located on the Tools > Manufacturing Services menu. For more information, see <i>Macro Editor</i> (on page 352).</p>
	<p>Check Manufacturability - Analyzes pipes in the current workspace and reports the objects that will be difficult or impossible to manufacture. This command is available on the Tools menu. For more information, see <i>Check Manufacturability</i> (on page 359).</p>

Selecting Objects

All objects in the Structural Manufacturing task have properties that you can edit. Using the **Select**  command on the vertical toolbar, you select the type of object that you want to edit or work with.



An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Plate Parts**, you can select only plate parts in a graphic view or in the **Workspace Explorer**.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

The Structural Manufacturing task includes these filters:

All Manufacturing Entities

Allows you to select any manufacturing object.

Assembly Margin

Limits your selection in a graphic view or in the **Workspace Explorer** to assembly margins.

Beam Par

Limits your selection in a graphic view or in the **Workspace Explorer** to beams.

Construction Graphics

Limits the selection of items to construction graphics.

Control Points

Limits your selection in a graphic view or in the **Workspace Explorer** to control points.

Detailing -- Bracket

Limits your selection in a graphic view or in the **Workspace Explorer** to brackets.

Detailing -- Collar

Limits your selection in a graphic view or in the **Workspace Explorer** to collars.

Detailing -- Feature

Limits your selection in a graphic view or in the **Workspace Explorer** to features.

Detailing -- Member

Limits your selection in a graphic view or in the **Workspace Explorer** to members.

Detailing -- Physical Connection

Limits your selection in a graphic view or in the **Workspace Explorer** to physical connections.

Detailing -- Plate

Limits your selection in a graphic view or in the **Workspace Explorer** to plate parts.

Detailing -- Profile

Limits your selection in a graphic view or in the **Workspace Explorer** to profiles.

Manufactured -- Member

Limits your selection in a graphic view or in the **Workspace Explorer** to manufacturing members.

Manufactured -- Plate

Limits your selection in a graphic view or in the **Workspace Explorer** to manufacturing plates.

Manufactured -- Profile

Limits your selection in a graphic view or in the **Workspace Explorer** to manufacturing profiles.

Manufacturing Tab

Limits your selection in a graphic view or in the **Workspace Explorer** to manufacturing tabs.

Margin -- Assembly

Limits your selection in a graphic view or in the **Workspace Explorer** to assembly margins.

Margin -- Part

Limits your selection in a graphic view or in the **Workspace Explorer** to part margins.

Marking

Limits your selection in a graphic view or in the **Workspace Explorer** to marking lines.

Pin Jig -- Assembly

Limits your selection in a graphic view or in the **Workspace Explorer** to assembly pin jigs.

Pin Jig -- Part

Limits your selection in a graphic view or in the **Workspace Explorer** to part pin jigs.

Shrinkage -- Assembly

Limits your selection in a graphic view or in the **Workspace Explorer** to assembly shrinkage.

Shrinkage -- Part

Limits your selection in a graphic view or in the **Workspace Explorer** to part shrinkage.

Tab

Limits your selection in a graphic view or in the **Workspace Explorer** to tabs.

Template

Limits your selection in a graphic view or in the **Workspace Explorer** to templates.

All

Allows you to select any object in a graphic view or in the **Workspace Explorer**.

**Inside**

Selects all objects entirely inside the fence.

**Inside/Overlapping**

Selects all objects entirely inside the fence and those objects outside but touching the fence at some point.























Selecting Multiple Manufacturing Objects
























When you have the filter set to **All**, Structural Manufacturing displays the **Select Mfg Parts** dialog box to help you narrow your selection.


















































You can select multiple items from the lists. If you select objects that have different attributes, then you can only edit the attributes that the objects have in common. Changes that you make are applied to all of the selected objects.

























Icons in the Workspace Explorer
























	Analysis Folder
	Analysis Model
	Assembly
	Assembly Connection
	Beam Part - undetailed
	Beam Part - detailed
	Beam Part - detailed, manufactured
	Bearing Plate - undetailed
	Bearing Plate - detailed
	Bearing Plate - detailed, manufactured
	Bearing Plate - group member
	Bearing Plate - group member, manufactured
	Bearing Plate - group master
	Bearing Plate - group master, manufactured
	Block
	Boundary Condition
	Bracket System or Leaf System
	Bracket Part - undetailed
	Bracket Part - detailed
	Bracket Part - detailed, manufactured
	Bracket Part - group member
	Bracket Part - group member, manufactured























	Bracket Part - group master
	Bracket Part - group master, manufactured
	Cable Nozzle on Equipment
	Cabletray Nozzle on Equipment
	Collar Plate - undetailed
	Collar Plate - detailed
	Collar Plate - detailed, manufactured
	Collar Plate - group member
	Collar Plate - group member, manufactured
	Collar Plate - group master
	Collar Plate - group master, manufactured
	Common Part Manager
	Conduit Nozzle on Equipment
	Coordinate System
	Coordinate System Axis
	Coordinate System Grid Plane
	Coordinate System Elevation Plane
	Coordinate System Radial Plane
	Coordinate System Radial Cylinder
	Curved Member Part - undetailed
	Curved Member Part - light
	Curved Member Part - light, manufactured
	Curved Member Part - detailed



















	Curved Member Part - detailed, manufactured
	Curved Member Part - group member
	Curved Member Part - group member, manufactured
	Curved Member Part - group master
	Curved Member Part - group master, manufactured
	Designed Equipment Component
	Designed Equipment
	Designed Handrail
	Designed Member - undetailed
	Designed Member - light
	Designed Member - light, manufactured
	Designed Member - detailed
	Designed Member - detailed, manufactured
	Designed Member - group member
	Designed Member - group member, manufactured
	Designed Member - group master
	Designed Member - group master, manufactured
	Edge Treatment
	Equipment
	Equipment Component
	Equipment Foundation
	Equipment Shape
	Equipment Solid

	Equipment Solid - Added
	Equipment Solid - Subtracted
	Equipment Solid - Suppressed
	Feature (includes corner feature, edge feature, face feature, slot, and so on)
	Footing
	Footing Components
	Foundation Port
	Frame Connection
	Handrail
	HVAC Nozzle on equipment
	Insert Plate - undetailed
	Insert Plate - detailed
	Insert Plate - detailed, manufactured
	Insert Plate - group member
	Insert Plate - group member, manufactured
	Insert Plate - group master
	Insert Plate - group master, manufactured
	Ladder
	Linear Member Part - undetailed
	Linear Member Part - light
	Linear Member Part - light, manufactured
	Linear Member Part - detailed
	Linear Member Part - detailed, manufactured
	Linear Member Part - group member

	Linear Member Part - group member, manufactured
	Linear Member Part - group master
	Linear Member Part - group master, manufactured
	Load Case
	Load Combination
	Member Assembly Connection Cutback Feature
	Member Assembly Connection Plate Part
	Member Assembly Connection
	Member Fireproofing
	Member Part
	Member Split Connection
	Member System
	Opening
	Panel - manufactured
	Physical Connection
	Pipe Nozzle on Equipment
	Planning Joint - butt weld
	Planning Joint - lap weld
	Planning Joint - tee weld
	Planning Joint Folder
	Plate - light
	Plate Part - undetailed
	Plate Part - light
	Plate Part - light, manufactured

	Plate Part - detailed
	Plate Part - detailed, manufactured
	Plate Part - detailed, manufactured, with manufacturing part assembly child
	Plate Part - group member
	Plate Part - group member, manufactured
	Plate Part - group master
	Plate Part - group master, manufactured
	Plate System or leaf system
	Profile Edge Reinforcement System or Leaf System
	Profile Edge Reinforcement Part
	Profile Edge Reinforcement Part - light
	Profile Edge Reinforcement Part - light, manufactured
	Profile Edge Reinforcement Part - detailed
	Profile Edge Reinforcement Part detailed, manufactured
	Profile Edge Reinforcement Part - group member
	Profile Edge Reinforcement Part - group member, manufactured
	Profile Edge Reinforcement Part - group master
	Profile Edge Reinforcement Part - group master, manufactured
	Seam - Design
	Seam - Intersection
	Seam - Planning
	Seam - Straking
	Slab Assembly Connection

	Slab
	Stair
	Standalone Beam Part - undetailed
	Standalone Beam Part - detailed
	Standalone Beam Part - detailed, manufactured
	Standalone Plate Part - undetailed
	Standalone Plate Part - detailed
	Standalone Plate Part - detailed, manufactured
	Standalone Plate Part - group member
	Standalone Plate Part - group member, manufactured
	Standalone Plate Part - group master
	Standalone Plate Part - group master, manufactured
	Standalone Profile Edge Reinforcement Part - undetailed
	Standalone Profile Edge Reinforcement Part - detailed
	Standalone Profile Edge Reinforcement Part - detailed, manufactured
	Standalone Profile Edge Reinforcement Part - group member
	Standalone Profile Edge Reinforcement Part - group member, manufactured
	Standalone Profile Edge Reinforcement Part - group master
	Standalone Profile Edge Reinforcement Part - group master, manufactured
	Standalone Stiffener Part - undetailed
	Standalone Stiffener Part - detailed
	Standalone Stiffener Part - detailed, manufactured

	Standalone Stiffener Part - group member
	Standalone Stiffener Part - group member, manufactured
	Standalone Stiffener Part - group master
	Standalone Stiffener Part - group master, manufactured
	Stiffener Part - undetailed
	Stiffener Part - light
	Stiffener Part - light, manufactured
	Stiffener Part - detailed
	Stiffener Part - detailed, manufactured
	Stiffener Part - group member
	Stiffener Part - group member, manufactured
	Stiffener Part - group master
	Stiffener Part - group master, manufactured
	Stiffener System or Leaf System
	Wall Assembly Connection
	Wall Part
	Wall Run
	Wall System

Icon Relationships

The following table describes the relationships indicated by the icons in the Workspace Explorer.

In general, there is a base icon for each type of entity. This base icon is then embellished depending upon the role that entity plays in the model. Using the System Plate Part as an example:



- Base icon for system plate parts. Base parts use blue icons



- Light (undetailed) system plate part. Light parts use white icons. Not all entities have this version.



- Light system plate part that has been manufactured. The plus symbol indicates that the part has manufacturing data. This only occurs if the detail part is deleted after manufacturing has been done.



- Detailed system plate part without manufacturing data. Detailed parts use red icons.



- Detailed system plate part with manufacturing data. The plus symbol indicates that the part has manufacturing data.

Common part grouping is also indicated by the icons. Under common part grouping, a master part is used as the basis for deciding commonality. Other parts that compare themselves to the master part are part of that common part group.

You can only select a grouped object if it is the master part. When you select the master part, all corresponding parts are also selected and manufactured. You cannot manufacture a group part separate from the master part.



- Detailed member of the common group. Member entities display a blue background.
























- Detailed and manufactured member of the common group.






























































- Detailed master part of the common group. Master entities display a yellow background.

































- Detailed and manufactured master part of the common group.

Group master, manufactured	Group master	Group Member, manufactured	Group Member manufactured, HasMFBearIn GPartAsAssembly Child	Detailed, manufactured	Detailed	Light, manufactured	Light	Base
Bearing Plate								
				N/A			N/A	
Bracket Part								
				N/A			N/A	
Collar Plate								
				N/A			N/A	

Insert Plate									
	N/A	N/A			N/A				
Standalone Plate Part									
	N/A	N/A			N/A				
System Plate Part									
									
Standalone Stiffener Part									
	N/A	N/A			N/A				



System Profile Edge Reinforcement Part									
					N/A				
Standalone Profile Edge Reinforcement Part									
	N/A	N/A			N/A				
System Stiffener Part									
					N/A				
Standalone Beam Part									
	N/A	N/A			N/A	N/A	N/A	N/A	N/A

System Beam Part									
	N/A	N/A			N/A	N/A	N/A	N/A	N/A
Linear Member Part									
					N/A				
Curved Member Part									
					N/A				
Designed Member									
					N/A				

Split Notification





When a seam is added, deleted, or modified, the plate or profile systems and parts split by the seam are affected. The software notifies different tasks to transfer attributes and objects to the new systems and parts. The Molded Forms, Structural Detailing, Planning, Hole Management, and Structural Manufacturing tasks are affected by split notification.

The software uses split notification when:

- A design or planning seam is added and **Execute Split** is run in the Molded Forms task.
- A design or planning seam that has already been split is deleted or modified in the Molded Forms task.
- An intersection seam is added and split when **Execute Split** is run in the Molded Forms task.
- A profile or plate system creating an intersection seam is deleted or modified in the Molded Forms task.
- A planning seam is added when **Manage Block Intersections**  is run and an intersecting part is set to **Split** or **Offset** in the Planning task.
- A planning seam is deleted when **Manage Block Intersections**  is run and an intersecting part is set to **Intersected** or **Assigned** in the Planning task.
- A planning seam is modified when the cutting plane of a block is modified in the Planning task.

Identifying Seams in Workspace Explorer

You can identify the type of seam in the **Workspace Explorer** by the icon.

-  Design seams
-  Planning seams
-  Straking seams
-  Intersection seams

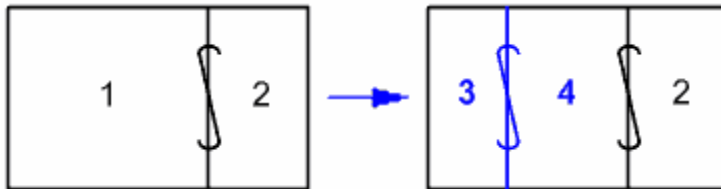
What do you want to do?

- *Add a seam* (on page [35](#))
 - *Delete a seam* (on page [37](#))
 - *Modify a seam when leaf systems are not added or removed* (on page [38](#))
-

Add a seam

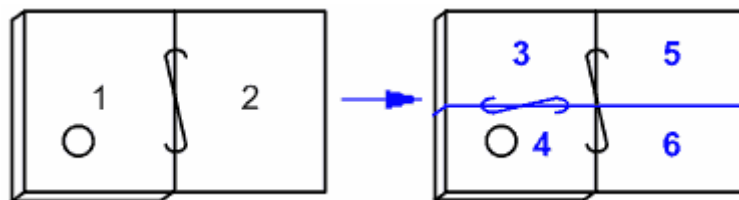
If you split a leaf system and its part with a new seam, the software creates new leaf systems and parts for each split, and deletes the original leaf system.

Molded Forms properties and material information are transferred to the new systems. The default naming rule is used to name the new leaf systems. Logical connections that cross the split are replaced by two new connections with the same properties.



If the part is detailed, Structural Detailing properties and features are transferred to the new parts. The software creates new assembly and physical connections using properties from the previous connections.

⚠ CAUTION Straking seams, the leaf parts created by straking seams, and free edge treatments are not transferred and are deleted.

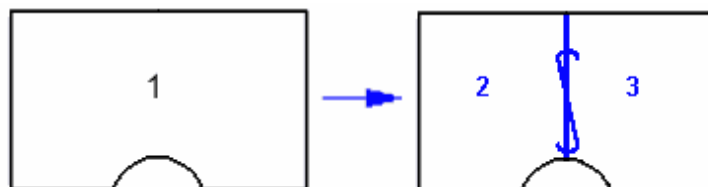


If a seam intersects an existing Structural Detailing feature, the feature is transferred to one of the parts.



Hole cuts and structural hole fittings in the Hole Management task are placed on the **To Do List**. Hole traces and outfitting catalog hole fittings are transferred to the new parts.

If a split passes through an edge feature definition point, the software creates two edge features. If the split does not pass through the definition point, then only the part containing the definition point contains an edge feature.



Block and assembly assignments in the Planning task are transferred. New parts are in the same assembly or block as the original part.

Manufacturing parts are updated using the **Manufacturing Service Manager**.

Connections to ladders, stairs, handrails, members, hangers, and equipment are transferred to the new part.

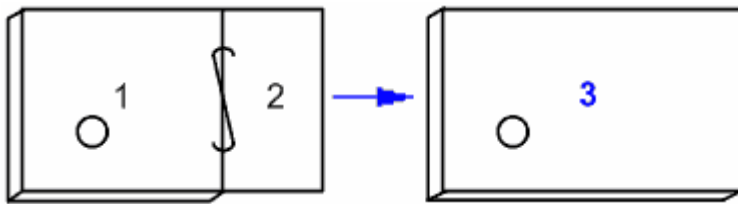
Delete a seam

If you delete or rebound a seam, the software deletes leaf systems and parts and creates new leaf system and parts.

Molded Forms properties and material information are transferred from the first system (as defined by the software) to the new system. The default naming rule is used to name the new leaf system. Logical connections that crossed the split are replaced by a new connection with the same properties as the first system.

Structural Detailing properties and features are transferred to the new part. If one part is detailed and the other is not, the new part is detailed.

⚠ CAUTION Straking seams, the leaf parts created by straking seams, and free edge treatments are not transferred and are deleted.



Hole cuts and structural hole fittings in the Hole Management task are placed on the **To Do List**. Hole traces and outfitting catalog hole fittings are transferred to the new parts.

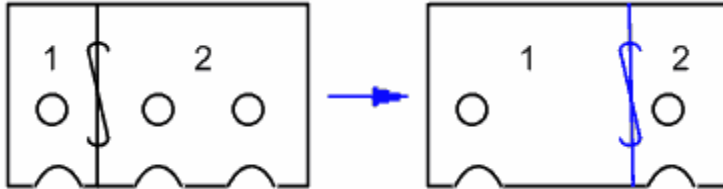
Block and assembly assignments in the Planning task are transferred. If the original parts were in the same assembly or block, then the new part is placed in that assembly or block. If the original parts were in different assemblies, then the new part is placed in the assembly last used by split notification.

Manufacturing parts are updated using the **Manufacturing Service Manager**.

Connections to ladders, stairs, handrails, members, hangers, and equipment are transferred to the new part.

Modify a seam when leaf systems are not added or removed

If you move a seam and the existing systems and parts are modified instead of replaced, then the software does not use split notification. **Structural Detailing** features are not transferred from one part to another, and are placed on the **To Do List**. If a feature is on one part and a seam is later modified so that it crosses the feature, the feature is still on one part. Split notification is not used when a seam is modified.



NOTES

- Split notification works with design seams from Molded Forms as well as with planning seams.
- If there are multiple features on one sketch, the software creates two sketches after the split; one for each part. Constraints associated with a sketched feature only go with one sketch after the split. You must manually add the constraints to the other sketch.

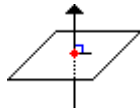
Plane Methods

You must often define planes used for object surfaces and sketching planes. Usually, but not necessarily, these planes are based from an existing plane or from a grid or reference plane created using the Grids task. The following methods define planes:

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points


Defines a plane using three points that you identify in the model.

What do you want to do?

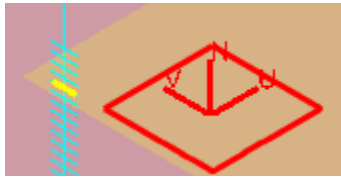
- *Define a coincident plane* (on page 39)
- *Define an offset plane* (on page 40)
- *Define a plane using a point and a normal vector* (on page 41)
- *Define a plane using three points* (on page 42)

- Move a defined plane

Define a coincident plane

1. Click **Offset from a Plane** .
2. In the model or **Workspace Explorer**, select a reference plane, a grid plane, or a planar plate system.

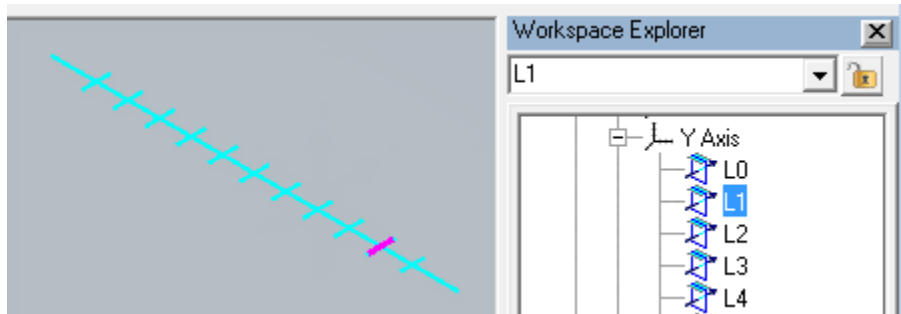
A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.



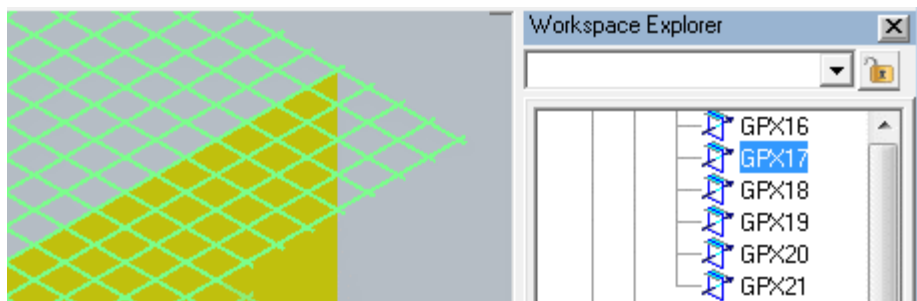
3. Check that **Offset Lock** is locked  and that the **Offset** value is 0.

NOTES

- You create reference planes in the Grids task by selecting **Ship** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can select reference planes in the **Workspace Explorer** or in a graphic view.




- You create grid planes in the Grids task by selecting **Grids** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can best select grid planes in the **Workspace Explorer**.

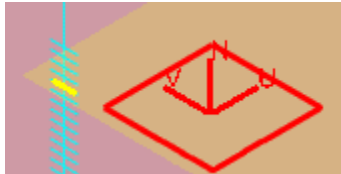


- For more information, see *Create Coordinate System (Grid Wizard)* in the *Grids User's Guide*.

Define an offset plane

1. Click **Offset from a Plane** .
2. In the model or **Workspace Explorer**, select the base plane from which to offset the new plane. The base plane can be a reference plane, a grid plane, or a planar plate system.

A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.



3. In the **Offset** box, type the offset distance from the plane and press TAB.

The plane moves to the new location.



-OR-

4. Graphically define the offset:

- a. Click **Offset Lock** .

The option changes to unlocked .


- b. Move the pointer to the required location and click.

*The plane and plane coordinate system move to the new location, **Offset Lock**  changes to locked , and the offset value displays in the **Offset** box.*


- c. If an adjustment to the offset is needed, type a new value in the **Offset** box.

5. To move the plane independent of its previous definition, see Move a defined plane.



TIPS

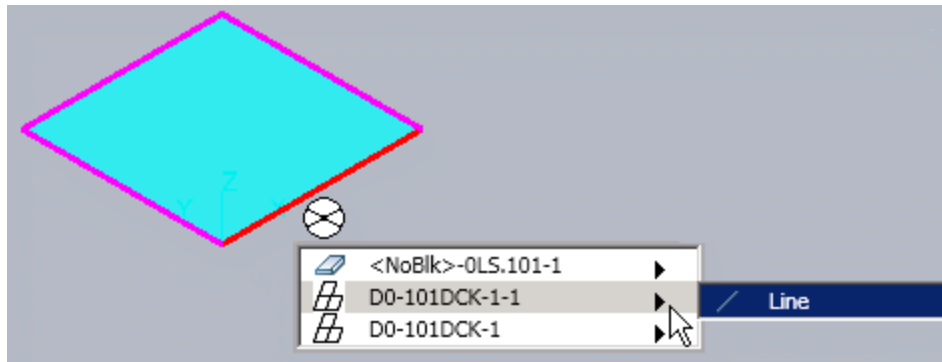
- You can graphically define the offset with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.
- By moving the pointer back and forth over the base plane in the graphic view, you can change the offset direction. You can also change the direction by changing the sign in the **Offset** box.
- When **Offset Lock** is set to locked , the **Offset** value does not change when the pointer moves.

Define plane using angle from plane

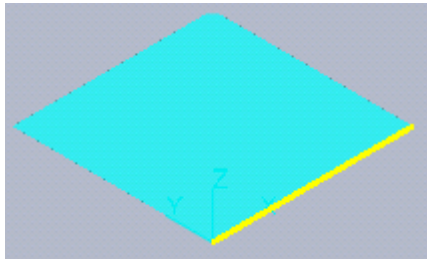
1. Click **Angle from Plane** .
2. In the model or **Workspace Explorer**, select the base plane from which to angle the plane that you are defining. This plane can be a planar plate system, face port of a plate port, or a grid plane.
3. Select a rotation axis. The rotation axis must be parallel to the surface of the selected plane.
TIP The rotation axis can be a linear seam, profile system, landing curve, system edge or connection.
4. In the **Angle** box, specify the rotation angle of the plane relative to the selected base plane.


Define a plane using a point and a normal vector

1. Click **Plane by Point and Vector** .
2. Select a vector  in the model that is normal to the required plane. The vector can be any linear geometry that is part of the construction of a model object, such as a plate system edge or a profile system landing curve. Use **QuickPick** to select a line geometry element.

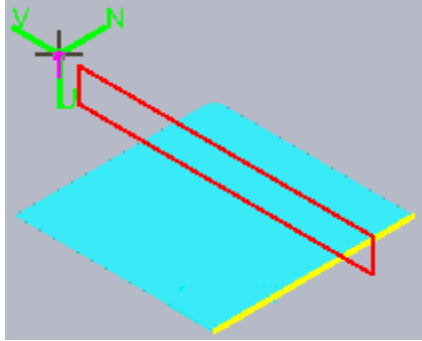


3. Click the **Line** geometry element.
The selected geometry highlights in yellow.



4. Move the cursor to a point  that is on the required plane, and then click.





A preview of the plane and its local U_V_N coordinate system displays. The plane is coincident with the point and normal to the vector.



5. To move the plane independent of its previous definition, see [Move a defined plane](#).

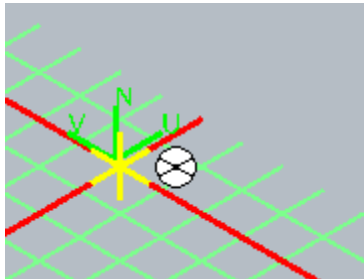
TIP You can graphically define the point with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.

Define a plane using three points

1. Click **Plane by Three Points** .
2. Specify on the first point  that defines the plane.
3. Specify the second point  that defines the plane.
4. Specify the third point  that defines the plane.

NOTES



- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools > Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.
- You can define points on grid planes, especially at intersections.





You can control the types of points using **Tools > Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

In Molded Forms, define parametric points by using **Insert > Topological Points**. For more information, see *Topological Points (Insert Menu)*.

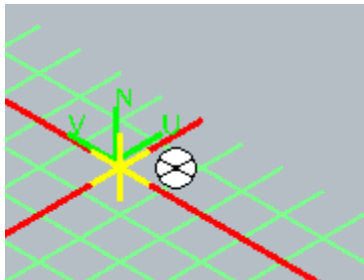
Define plane by vectors normal

1. Click **Plane by Vectors Normal** .
2. In the model or **Workspace Explorer**, click the plane  to which the new plane is normal.

TIP The plane can be a planar plate system, a face port of a plate part, or a grid plane.
3. Specify the first point  that defines the vector.
4. Specify the second point  that defines the vector.

NOTES

- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools > Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.
- You can define points on grid planes, especially at intersections.



You can control the types of points using **Tools > Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

- In Molded Forms, define parametric points by using **Insert > Topological Points**. For more information, see *Topological Points (Insert Menu)*.


Boundary Methods

You must often define boundaries when creating plate systems, profiles, edge reinforcements, and other objects. The common methods for defining boundaries are:

Pick Boundaries

Define boundaries by selecting them graphically either in a graphics view or in the **Workspace Explorer**.

Pick Edges

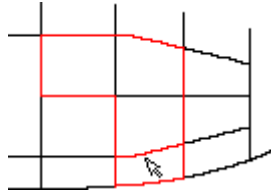
This option is only available if you pick a plate as a boundary. Select the plate edges to use. Click **Pick Boundaries**  to exit the **Pick Edges** option and select other boundaries.

Boundary List

Define boundaries by selecting them from a list of potential boundaries. For more information, see *Boundary List Dialog Box* (on page 47).

Boundary Curve

Define the boundary by drawing its shape.




Solve Ambiguity


If the boundaries that you selected can form the object in more than one way, then you have defined an ambiguous solution. The software prompts you to select one or more solutions.

What do you want to do?





- *Pick boundaries* (on page 44)
- *Use the Boundary List dialog box* (on page 44)
- *Define a boundary curve* (on page 45)
- *Solve ambiguous solution created by selected boundaries* (on page 45)
- *Rebound objects* (on page 46)

Pick boundaries

1. Click **Pick Boundaries** .
2. In a graphic view or **Workspace Explorer**, select an object or grid plane to use as a boundary.
3. Continue selecting objects or grid planes until you have completed defining the boundaries for the object.





 **NOTE** To remove an object from the group of boundaries, select the object again.

Use the Boundary List dialog box

1. On the command ribbon, click **Boundary List** .
The **Boundary List** dialog box displays.
2. To remove a boundary, select the boundary in the list, and then click **Remove** .
3. To remove all boundaries, click **Remove All** .
4. To add a new boundary, click **Add** , and then select a boundary in the graphic view or in the **Workspace Explorer**. You can also type the boundary name and an optional offset value in the **Offset** box or the **Name** box.

 **NOTE** For more information, see *Boundary List Dialog Box* (on page 47).

Define a boundary curve

1. Select **Boundary Curve** .
2. Define the sketching plane for the curve.
 - Define a coincident plane* (on page 39)
 - Define an offset plane* (on page 40)
 - Define plane using angle from plane* (on page 41)
 - Define a plane using a point and a normal vector* (on page 41)
 - Define a plane using three points* (on page 42)
 - Define plane by vectors normal* (on page 43)
3. Click **Add Intersecting Item** .
4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the 2D environment.
5. Click **Add Projection Item** .
6. Select objects in the model that do not intersect the sketching plane and that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the 2D environment.
7. Select **Auto** to automatically add all related objects to the select set, if needed.
8. Click **Sketch 2D** .



The 2D environment appears.
9. In the 2D environment, sketch the boundary.

NOTE You can constrain the boundary to geometric construction points referenced from profile cross-section key points or offset from plate systems. For more information, see *Point at Offset from Keypoint Command* and *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the 2D environment.
10. Click **Finish** in the 2D environment.

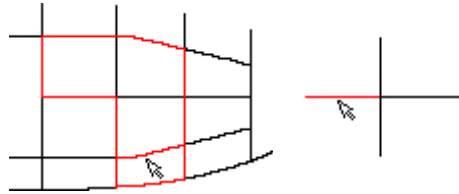
The 3D environment and the main command ribbon return.

NOTE Deleting a boundary curve from the 2D environment also removes it from the **Boundary List** dialog box.


Solve ambiguous solution created by selected boundaries

The command switches to **Solve Ambiguity**  automatically if an ambiguous solution exists after clicking **Accept**  or **Finish**. All possible solutions created by the selected boundaries appear in the graphic view.

1. Move the pointer over a solution, and then click to select it.
2. Continue to select solutions until you have defined the entire bounded object. A correct solution to ambiguity meets these conditions:
 - Each solution must have a common edge with at least one other solution.
 - Each boundary must be used by at least one solution.



NOTES

- To modify the ambiguous solution after creating an object, select **Solve Ambiguity** , and select new solutions.
- To remove a previously-selected solution, select the solution again.
- Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Rebound objects

Use this procedure to replace a boundary with another boundary, such as when the boundary is used by multiple objects.


1. Select **Tools > Automatic Reconnect**.

*The **Automatic Reconnect** dialog box displays.*

2. On the **Boundary** tab, select **Rebound**.
3. For **Old boundary filter**, select an object type from the list. For example, if the boundary is a reference plane, select **Grids**.
4. Specify the old boundary object in the **Old boundary object** box. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box, or select an object from the list, which shows the last ten selections you have made.
5. For **New boundary filter**, select an object type from the list. For example, if the boundary is a plate system, select **Plate Systems**.
6. Specify the new boundary object in the **New boundary object** box. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box, or select an object from the list, which shows the last ten selections you have made.
7. Optionally, to further filter restrict the objects to rebound, select values for **Scope filter** and the **Entity filter**.
8. Click **Apply**.
*All entities that are bounded to the old boundary display in **Candidates list** and highlight in the graphic view.*
9. In **Candidates list**, clear the check box for objects you do not want to rebound.
10. Click **Rebound**.
11. Click **View Log** to see the rebound status.
12. Click **Close**.

Boundary List Dialog Box

Displays the boundaries for the object that you are placing:

- Model objects, such as plate systems or profile systems.
- Reference planes and other plane definitions.
- Sketched boundary curves created using the **Sketch 2D**  option on a supported ribbon. Boundary curves are prefixed with **SketchGroup_**.
- A standalone plane created during a copy, move, or mirror operation when a boundary was not included in the operation. You usually want to replace the standalone plane with a new boundary.

Open this dialog box by clicking **Boundary List**  on the command ribbon during the boundary definition step.

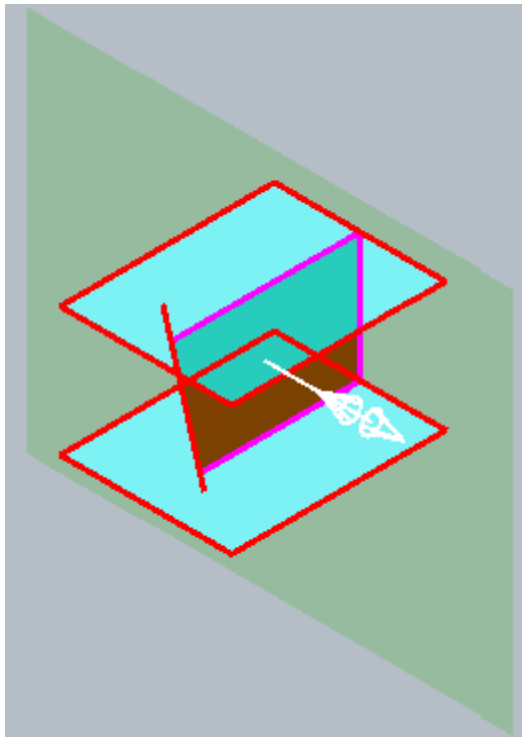
Preview Type

You can press ARROW DOWN ↓ and ARROW UP ↑ to step through the boundary list. Each selected boundary highlights in the graphic view according to the selected preview type.

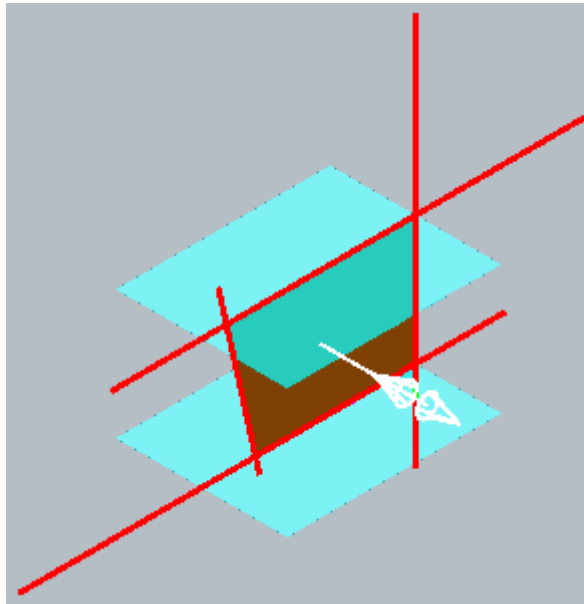
Tools > Options defines the **Highlight** color. For more information, see *Colors Tab (Options Dialog Box)* in the *Common User's Guide*.

In the following examples, two plate systems, a reference plane, and a sketched boundary are the selected boundaries of the vertical plate.

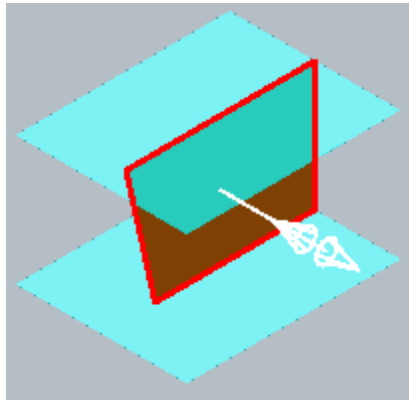
Bounding Objects - Highlights the selected boundary object as an outline. Reference planes display as translucent shaded planes.




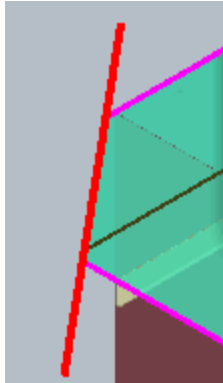
Bounding Geometry - Highlights the internal boundary geometry used by the software. The internal geometry is typically an unlimited length line for model objects and reference plane boundaries.



Edges - Highlights edges of the bounded object where the bounded object intersects with the selected boundary.



NOTE For **Bounding Objects** and **Bounding Geometry**, the complete sketched boundary curve geometry displays as drawn in **Sketch 2D** , as shown in the following example.



Add

Specifies a new boundary. You can select a boundary in the graphic view or the **Workspace Explorer**.

NOTES

- You can also type the boundary name and an optional offset value in the **Name** box or the **Offset** box, and then press TAB or ENTER.
- Invalid text displays with a gray background in the **Name** box or the **Offset** box.



Remove

Removes the selected boundary from the list.

NOTE Deleting a boundary curve from the 2D environment (**Sketch 2D**  on the main command ribbon) also removes it from the **Boundary List** dialog box.



Clear All

Removes all boundaries from the list.



Disconnect

Removes the relationships of a boundary to its definition method. When you disconnect a boundary, it becomes a standalone plane. This option is active by default for a standalone plane created during a copy, move, or mirror operation.

NOTE After disconnecting a boundary, you must select a replacement boundary in the graphic view or the **Workspace Explorer**.



Extend Sketch Boundary in Graphic View

Extends a sketched boundary to a point in space or to another element selected in the graphic view. This option is available when you select a sketched boundary (with a name prefixed with **SketchGroup_**).

Name

Specifies the name of a boundary.

Offset

Specifies an offset distance from the boundary. You can type the following formats for offset values:

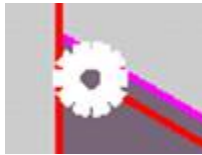
- Offset distance without units.
Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.
- Offset distance with units. The value converts to the default units.
Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.
- Coordinate System: Frame plus or minus an offset distance.
Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**
Example (when the model has one coordinate system): **F20 + 1.5 m**

NOTE Invalid text displays with a gray background in the **Name** box or the **Offset** box.

Apply

Displays a preview of the boundaries in the graphic view.

NOTE Gaps between boundaries display with a glyph.



The glyph uses the **Handle** color defined in **Tools > Options**. For more information, see *Colors Tab (Options Dialog Box)* in the *Common User's Guide*.

OK

Closes the dialog box and saves the boundary changes.

NOTE To cancel boundary changes, you must click **OK** and then exit the parent command without clicking **Finish**.

See Also

Use the *Boundary List dialog box* (on page 44)

Permission Group Behaviors Between Tasks

Permission groups provide controls for when, where, and by whom the model can be updated. Permission groups are assigned according to several common types of workflow. They typically fall into the following categories:

- By Function: such as Outfitting versus Structure, or Piping versus Equipment versus Molded Forms versus Structural Detailing.
- By System: such as Hull versus Deck1 versus TransvBhd1. This includes the ability to place the hull system (and its child leaf systems, seams, connections related seams, reference curves and openings) in a different permission group than the hull detailing objects and other connected objects.
- By Area: Fwd versus Aft, or by block or assembly.

- Combinations: for example, By Function and By Area (Molded Forms for Block1 versus Detailing for Block1).

The permission group of an object is not modified regardless of the **Active Permission Group** unless the permission group is explicitly changed on the **Configuration** tab of the **Properties** dialog box as described under the change propagation sections below.

Molded Forms Behavior

Plate systems and profile systems are assigned to the **Active Permission Group**.

Leaf systems, light parts, reference curves, and logical connections are assigned the permission group of their parent.

Seams and openings are assigned to the permission group of the plate or profile system they cut. You must have write access to the root plate system being cut when placing an opening or design seam.

In the **Execute Split** command, seams and knuckles that you do not have permission to modify are not selectable in the table.

Logical connections between profile systems and plate systems are in same permission group as the profile systems.

Logical connections between different root systems are created as children of the bounded or penetrated system.

Summary of Molded Forms permission behaviors:

Object	Permission Group	Comments
Root plate system	Active Permission Group	
Leaf plate system	Same as root plate system	
Root and child logical connections	Same as dependent root plate system	
Reference curve	Same as root plate system	
Design seam, intersection seam	Same as root plate system	Must have write access to root plate system
Opening	Same as root plate system	Must have write access to root plate system
Manual logical connection	Permission group of system for which you have write access, use first system selected if you have write access to both systems	Must have write access to at least one of the systems being connected
Root profile system	Active Permission Group	Profiles are handled similarly to plates

Object	Permission Group	Comments
Logical connection between stiffener or edge reinforcement and the plate being stiffened	Same permission group as root profile system	
Beam	Active Permission Group	

Molded Forms Change Propagation

The root system and its child leaf system, logical connections, reference curves, seams and openings always have the same permission group. When the permission group is changed for any of these objects, the permission group is also changed for the other objects.

If plate system A is bounded by plate system B, you can modify A without write access to B. Plate System B is placed on the **To Do List** as **Out of Date** if the change to A impacts B.

Structural Detailing Behavior

The detailed part permission group is not assigned from the parent system permission group. Detailed parts created with **Execute Detailing** are assigned to the **Active Permission Group**. You can modify this assignment. You must have write permission to the part to detail the part.

If a detailed part is deleted, the resulting light part maintains the permission group of the detailed part.

You can assign different permission groups to parts sharing the same leaf system as created by a planning seam.

You can only add seams and features to a part if you have write permissions to the part.

Summary of Structural Detailing permission behaviors:

Object	Permission Group	Comments
Light Root Plate Part (before detailing)	Same as plate system at creation	
Detailed Root Plate Part	Active Permission Group when detailed. Part keeps this permission group if the detail part is deleted.	Must have write access to the light part
Leaf part	Same as root part	
Straking seam	Same as root part	Must have write access to root part
Feature placed by command (edge, corner, sketched, free edge treatment)	Same as root part	Must have write access to root part

Object	Permission Group	Comments
Assembly Connection	Same as dependent part	
Feature, part, physical connection created by assembly connection	Same as assembly connection	
Free end cut object and its features	Same as root part	
Standalone part	Active Permission Group	

Structural Detailing Change Propagation

The detailed part, its child leaf parts, features and straking seams always have the same permission group. Assembly connections always have the same permission group as the dependent part in the connection. Assembly connection children always have the same permission group as the assembly connection. Standalone parts plus associated features and connections always have the same permission group. When the permission group is changed for any of these objects, the permission group is also changed for the other objects.

If you do not have write access to a part, the part is placed on the **To Do List** as **Out of Date** if a change is made to the part. Assembly connections and features are also not created until the part is updated on the **To Do List** by someone with write access.

Hole Management Behavior

Hole traces and outfitting catalog hole fittings are assigned to the **Active Permission Group**.

Hole cuts and structural hole fittings are assigned the permission group of their parent structural part (plate or profile).

Summary of Hole Management permission behaviors:

Object	Permission Group	Comments
Hole trace	Active Permission Group	
Hole cut	Same as root plate part	Must have write access to parent plate part. Root plate part must be detailed in the Structural Detailing task.
Outfitting catalog hole fitting	Active Permission Group	
Structural hole fitting	Active Permission Group	Must have write access to parent plate part. Root plate part must be detailed in the Structural Detailing task.

Hole Management Change Propagation

The permission group of an outfitting catalog hole fitting can be changed without changing the permission group of the hole trace.

The permission group of a structural hole fitting can be changed without changing the permission group of the hole cut or parent plate part.

A hole cut and its parent plate part always have the same permission group. When the permission group is changed for one of these objects, the permission group is also changed for the other object.

Planning Behavior

Planning permission group behavior is related to Molded Forms and Structural Detailing behavior because planning seams split leaf systems.

When **Manage Intersections** is run and an intersecting part is set to **Split** or **Offset**, planning seams are created in the **Active Permission Group**. You must have write access to the root system.

Summary of Planning permission behaviors:

Object	Permission Group	Comments
Planning seam	Active Permission Group	Must have write access to the root system of the parts being split.
Block or assembly	Active Permission Group	

Planning Change Propagation

Write access is required to both parts to remove a planning split using **Flatten Assembly** or **Manage Intersections**. The resulting new part is assigned to the same permission group as the original parts if they belonged to the same permission group. The new part is assigned to the **Active Permission Group** if the original parts were in different permission groups.

Global Workshare Behavior

Global Workshare allows managed replication of the model at several sites. One of the restrictions of a Workshare configuration is that you only have Read access to objects that are controlled by non-local permission groups.

A system is a logical grouping of sub-systems. When you add or remove a sub-system, you are modifying the definition of the parent system. Therefore, you must have write access to the parent system. You do not need write access to the grandparent system.

When participating in a Global Workshare Configuration, you must manage all permission groups at the host site. The sub-system requirement to have write access to its parent system is not possible if the sub-system's permission group is created at the satellite site and the parent system's permission group is created at the host site.

For example, your host site is Houston and your satellite site is London. You create a system named "Midbody", and its controlling permission group is in Houston. You assign John, who works in London, write access. During the workshare replication process, the Midbody system and

permission group is replicated in London. In London, John can add systems, plates, profiles and whatever else he wants to the Midbody system because John was given write access to the system's permission group in Houston. John cannot delete or change any of the properties of the Midbody system in London because the host site, Houston, owns it. He can only add objects to the system. If John were to travel to Houston and log on there, he could delete or change any of the properties of the Midbody system because the Houston host site owns it.

Permission Group Behaviors in Structural Manufacturing

When a new Structural Manufacturing object is created, or the permission group and the approval status of an existing Structural Manufacturing object is modified, then the permission group and approval status is propagated and applied to any children under the parent object. For example, the children of a given plate or profile maintains the same permission group and approval status as the parent object.

Assembly Margin

Inherits the permission group and approval status of the parent assembly.

Fabrication Margin

Created with the active permission group.

Pin Jig

Inherits the permission group and approval status of the parent objects. They pass these settings on to the pin jig children, such as intersection points, remarking lines, and so on.

Shrinkage

Inherits the permission group and approval status of the parent object.

Marking Lines Folder

Inherits the permission group from the parent of the object to which the marking lines belong.

Marking Lines

Inherits the permission group and approval status of the parent object.

Templates

Inherits the permission group and approval status of the parent objects. They pass these settings on to the template children, such as base control lines, sketching lines, and so on.


Manufacturing Plate

Inherits the permission group and approval status of the parent object.

Manufacturing Profile

Inherits the permission group and approval status of the parent object.

Sketch 2D Best Practices

The **Sketch 2D**  option is used in several commands to create sketched boundaries, openings, curves for extruded plate systems, and profile and seam landing curves. The following Sketch 2D best practices help you create objects in Molded Forms that you detail in the Structural Detailing task and manufacture in the Structural Manufacturing task.

Templates

Sketch 2D default display options are saved in the file *[Product Folder]\Common2D\Symbol2D\Templates\template.sha*. You can change the default display by changing options in this file:

1. Right-click *template.sha* and select **Properties**.
2. Clear **Read-only** and click **OK**.
3. Open *template.sha* and change any of the following default display options:
 - **File > Properties**. You can change the units displayed in command ribbon bars on the **Units** tab.
 - **View > Toolbars**. You can select the toolbars that are displayed.
 - **Format > Dimensions**. You can change the units used in dimensions, the text size, and other dimension properties.
 - **Tools > Options**. You can change the default colors on the **Colors** tab.
 - **Tools > Display Manager**. You can change the default geometry color, line type, and width for individual layers.
 - **Tools > SmartSketch Settings**. You can select the SmartSketch relationships that are enabled on the **Relationships** tab.
4. Save and close the file.

Fractional Units

The default units and precision values in Sketch 2D are defined in a 3D task. For example, if the file *[Product Folder]\Common2D\Symbol2D\Templates\template.sha* has default units set to **mm**, and the units of measure in Molded Forms are set to **ft-in**, Sketch 2D defines the units of measurement as **ft-in**.

The **Properties** dialog box in Sketch 2D allows you to view the defined units of measurement and to specify the fractional precision value. If you need to use a specific fractional precision value for all tasks in Sketch 2D, you can specify the precision value in the template file, which takes precedence over the precision value defined by the 3D task.


The following example shows how to modify the length readout precision to **1/64** in the template file:




1. In the *template.sha* file, set the precision value to **1/64**, and save the file.
2. In the Molded Forms task, click **Tools > Options**.
*The **Options** dialog box displays.*
3. Under **Units of Measure**, set the distance readout to **ft-in (fractional)**.
4. Click **OK**.

*The distance units in Sketch 2D are now in **ft-in** with a precision value of **1/64**.*

Number of sketch files allowed per plate

Sketch 2D geometry for an object is saved in 2D files cached in the model and related to the object.

The **Boundary Curve**  step for the plate system commands uses one 2D file. All boundary curves are in one 2D file for each plate system, and are modified, added, or deleted in the same file.

The **Place Opening** , **Stiffener Profile System by 2D Projection** , and **Design Seam by 2D Sketch**  commands can each have:

- Multiple 2D files, with a separate file for each object on one plate system. The objects are modified in the 2D file, and deleted from the graphic view or **Workspace Explorer**.
- Geometry for multiple objects in one 2D file. The objects are modified or deleted in the 2D file.
- One 2D file that applies to multiple plate systems. Changes to the 2D file are reflected on all plate systems.
- A combination of these files.

Add items to sketch

The **Add Intersecting Item**  and **Add Projection Item**  steps are used to add reference structure to Sketch 2D. Select the minimum number of needed items.

NOTE More than 30 items added to a sketch file can slow down software performance when the object is recomputed.

Next, you can select **Auto** to automatically add all objects that are relative to the object to be sketched to the current select set. Items that pass the criteria are added to the select set and are highlighted in the graphic view, as well as the **Workspace Explorer**.

The filter criteria that the software uses to determine eligible candidates depend upon whether **Add Intersecting Item** or **Add Projection Item** was selected in a previous step. The following table lists the various relevant objects that are automatically imported based on the target, or base sketch, object type.

Base Sketch Object	Intersecting Objects	Projecting Objects
Plate System	<ul style="list-style-type: none"> ▪ Stiffeners ▪ Edge Reinforcements ▪ Seams ▪ Bounded Objects ▪ Bounding Objects ▪ Knuckle Reference Curves 	<ul style="list-style-type: none"> ▪ Reference Curves ▪ Stiffeners ▪ Penetration Plates
Plate Part	None supported in the current version of the software	None supported in the current version of the software
Profile System	None supported in the current version of the software	None supported in the current version of the software
Member	None supported in the current version of the software	None supported in the current version of the software
Designed Member (Built Up)	Same as Plate System	Same as Plate System



If no objects are added to the select set, the software displays a message on the status bar.

If you clear **Auto**, all selected objects, whether added to the select set automatically or manually, are cleared. Selecting **Auto** adds back those objects that initially passed the filter criteria, but it does not add back any objects that you selected manually.

⚠ CAUTION When the selected plate system is large, **Auto** automatically adds all objects relative to the sketch in the context of *intersecting* or *project*. This can slow down software performance.



Layers

The software automatically defines several layers in Sketch 2D. Reference items (selected automatically by the software or manually using the **Add Intersecting Item**  and **Add Projection Item**  steps) are placed on those layers:

Default Layer

The software only uses geometry that you create on the **Default** layer. You can put other geometry, such as construction lines, on other layers. The Default layer must be the first layer alphabetically. To avoid a conflict, you should add a "z" prefix to any layer which you add, such as "z_construction."

Infinite Elements Layer

Contains reference items with infinite geometry intersecting the sketch plane, such as grid planes. Objects on this layer can be used with dimensions and relationships.

Inputs Layer


Contains other reference items intersecting the sketch plane that can also be used with dimensions and relationships. The reference geometry includes openings, design seams, plate systems, and profile systems.

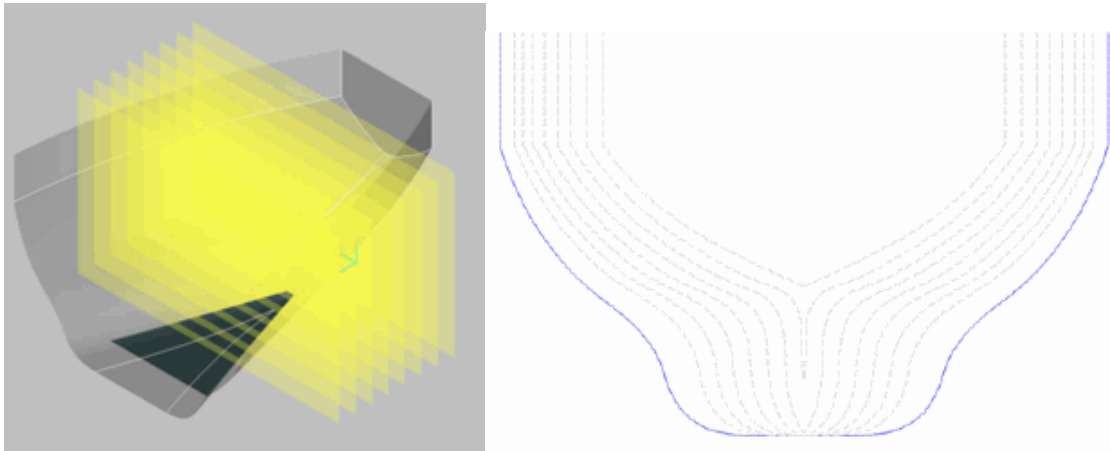
No_Constrained_Elements Layer

Contains reference items that do not intersect the sketch plane. Also contains reference items that cannot be used with relationships and dimensions, such as plate and profile parts. By default, you cannot select reference geometry on the **No_Constrained_Elements** layer. To make the geometry selectable, you must make **No_Constrained_Elements** the active layer, and then switch back to the **Default** layer.

You can add relationships and dimensions to reference geometry on the **No_Constrained_Elements** layer, but the relationships are not saved when you click **Finish**.

Multiple reference planes

You can add reference planes to Sketch 2D that are different than the sketching plane, but parallel to the sketching plane. This is useful in defining landing curves for projected profiles and seams on the hull. Use **Add Projection Item**  to add the reference planes.



The additional reference plane intersection geometry is gray in color and placed on the **No_Constrained_Elements** layer.

Using SmartSketch Drawing Editor to save sketch geometry

SmartSketch Drawing Editor, delivered to *[Product Folder]\Shape2D\Bin\shape2dserver.exe*, can be used to save geometry commonly used in Sketch 2D, such as:

- Mother curves and extrusion curves for extruded plate systems.
- Boundary curves.
- Openings or sketched features not included in the Catalog.
- Landing curves for projected profiles and seams.





You can copy the geometry from Sketch 2D and paste it into SmartSketch Drawing Editor, or create the geometry directly in SmartSketch Drawing Editor. Geometry can be saved in multiple SmartSketch Drawing Editor files or into a single file with multiple sheets. The geometry can be copied from SmartSketch Drawing Editor and pasted into Sketch 2D as needed.

Deleting reference geometry

You can delete reference geometry from the sketch when the software includes geometry that you do not want to see. The reference structure is not deleted, and the reference geometry is recreated the next time that you open the sketch.


Modifying groups of geometry

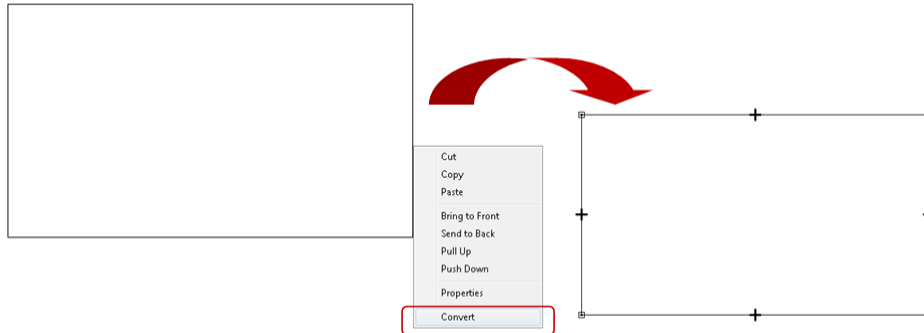
The software automatically groups Sketch 2D geometry when you click **Finish**. When you modify the existing 2D geometry of an object, you must maintain the original group identifier by using the following methods:

1. To modify or delete existing geometry, use **QuickPick**, **Top Down**  or **Bottom Up**  with **Select Tool**  to select the geometry instead of the group.
2. To add new geometry to a group, use **Edit > Group Modify**.
3. To replace all geometry in a group with geometry copied to the clipboard, use **Replace Group Elements** .

Rectangles

Use **Line**  or **Line/Arc Continuous**  to create rectangles.

If you use **Rectangle** , you must select **Tools > Maintain Relationships**. Right-click the geometry, and then select **Convert**. This converts the rectangle into four lines and the correct connection relationships are created at the corners.



Constraints

Use relationships and dimensions to constrain geometry to reference structure. This keeps the geometry properly related to the reference structure if the reference structure changes.

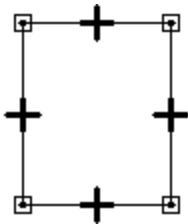
Keep relationships and dimensions as simple as possible. This allows the geometry the best chance to update if the reference structure changes.

Connections

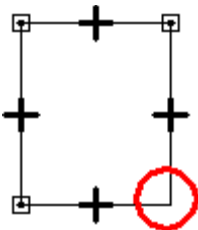
Connection relationships are required for the software to create connected geometry.

Do not create a connection between the corner of the sketched geometry, and the corner of the bounding object. Instead, move the constraint slightly off the corner.

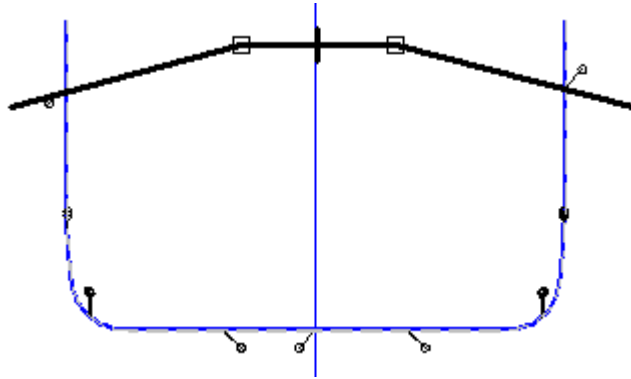
Good: All endpoints of opening are connected.



Bad: Missing connection at one corner causes the creation of the opening to fail.



Good: All endpoints of extrusion curve are connected.



Bad: Missing connection on extrusion curve.

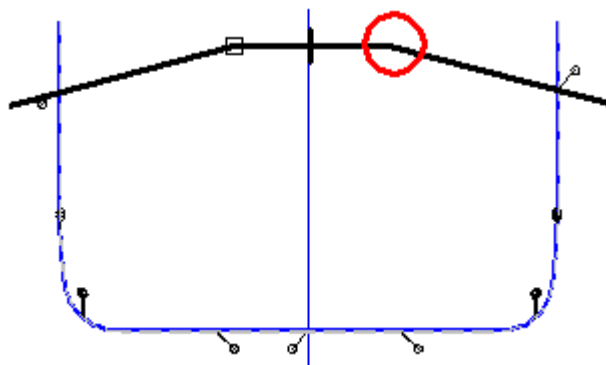
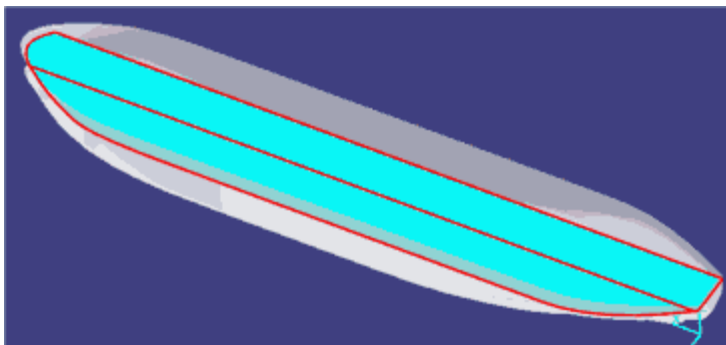


Plate system is missing part of the curve.

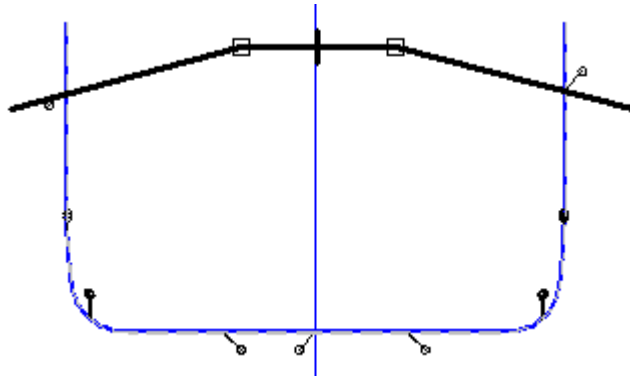


Extending sketch geometry to intersect bounding structure

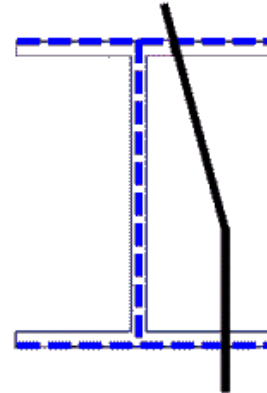
Extend sketch geometry beyond bounding reference structure geometry to ensure that the sketch geometry intersects the reference structure at all locations.

Good:

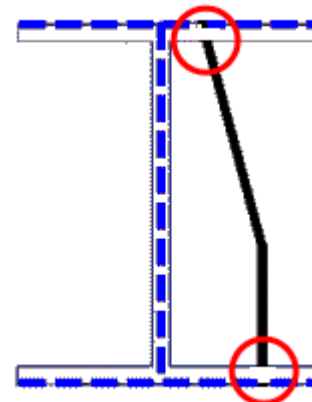
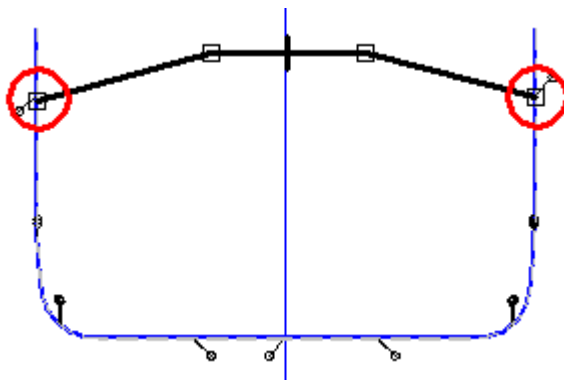
Extrusion curve extends beyond the hull.



Bracket boundary curve extends beyond the top flange and the bottom of the bottom flange.

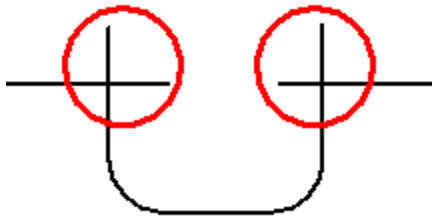


Bad:



Overlapping sketch geometry for boundary curves

A boundary curve with all geometry connected is treated as one edge in the Graphics View. You may sometimes want parts of the curve to be treated as separate edges at sharp discontinuities such as corners. This allows you to add edge reinforcements in the Molded Forms task or edge treatments in the Structural Detailing task to part of the boundary curve without defining additional boundaries. In order to separate the boundary curve into separate edges, overlap the boundary curve geometry instead of connecting it at the corners:

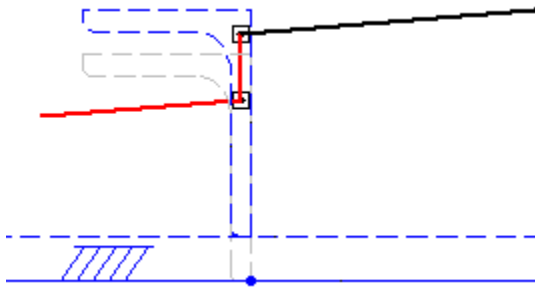


Creating boundary curves on the thickness side of a plate

If your plate boundaries include a plate system with the thickness direction towards the plate that you are creating, a profile mounted to the bounding plate system and a sketched boundary, and then the boundary curve must intersect both the profile system and profile part.

In the following example:

- The solid blue line is the molded surface of the bounding plate system.
- The dashed blue line is the thickness direction for the bounding plate part.
- The dashed gray profile outline is the bounding profile system mounted to the bounding plate system at the molded surface (on the **No_Constrained_Elements** layer).
- The dashed blue profile outline is the bounding profile part mounted to the thickened bounding plate part.
- The solid black line is the intended boundary curve intersecting the profile part.
- The solid red line is the required adjustment to the boundary curve, passing through both the profile part and profile system, and intersecting the profile system.

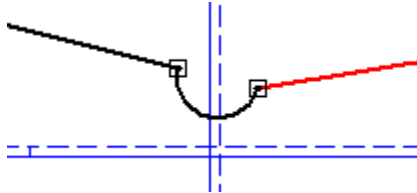


The black line is the intended boundary curve. It can be constrained to the dashed blue profile part using relationships and dimensions. The red lines are additional geometry that must connect to the black boundary curve, pass through the profile part geometry, and intersect the dashed grey profile system geometry.

Arcs in boundary curves

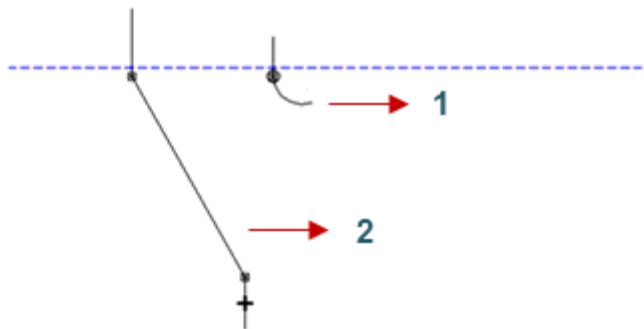
If a boundary curve ends with a circular or elliptical arc, you must add a line segment pointing away from the direction of the arc. This prevents the arc from extending back onto the boundary curve as the curve is calculated.

In the following example, the black curves are the intended boundary curve. The red line is the extension to the boundary curve that prevents the arc from extending back onto the boundary curve.



Multiple curves

Do not create a close boundary curve contour that overlaps trimming input geometry. Instead, draw two separate curves. In the following example, there are two separate curves, one for corner feature and one for diagonal boundary.




1 - Corner feature curve

2 - Diagonal boundary curve

Associative offsets

The **Associative Offset** command allows you to create a curve based on an existing curve, and place the new curve at a specified offset. To use this command, you must add it to a toolbar.

1. Right-click an existing toolbar, and click **Toolbars**.
2. Click **Customize** on the **Toolbars** dialog box.
3. Click **Manipulation** on the **Categories** list.
4. Drag the **Associative Offset**  icon from the **Buttons** list to an existing toolbar.

For more information on the **Associative Offset** command see the **Associative Offset Command** topics in the *SmartSketch Drawing Editor* help file.

SECTION 2

Marking Lines

Most markings are automatically applied to the detailed parts by rules. However, a certain amount of markings need to be placed manually.

NOTE When you create or edit a marking line, the dialog box title displays the plate part name using the format **Plate Part:** *<name of the plate part>*.

 **untitled.ses : Structural Detailing : Plant : Filter - [Plate Part : <NoBlk>-OLS.2-1]**

The Structural Manufacturing task provides the following commands to create marking lines:



Marking - Creates the manual marking lines on plate parts, profile parts, members, and pin jigs. For more information, see *Marking* (on page 65).



Marking by Definition - Creates the manual marking lines based on symbol definition. For more information, see *Marking by Definition* (on page 89).

Marking



Creates the manual marking lines on plate parts, profile parts, members, and pin jigs. Marking lines are used to assist the construction processes.

Marking Line Types

Location

The trace curve of plate, profile, collar plate, and bracket. These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, curved profile, template, jigs and remarking objects.

End connection

The cross section stamp on a plate or profile. These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Lap connection

These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Tab and bridge

A marking for the final cut. These marking lines are created automatically. They are applied to flat plate and curved plate objects.

Template line

Template positions on a plate. These marking lines are created automatically. They are applied to curved plate objects.

Roll boundaries

Roll lines showing roll range with radius and direction information. These marking lines can be created automatically or manually. They are applied to curved plates and curved profile objects.

Flat of bottom, flat of side, and other hull knuckle lines

Delivered by the naval architecture software through the Molded Forms task. These marking lines can be created automatically or manually. They are applied to curved plates and template objects.

Knuckle line

Handles corrugated plates. These marking lines can be created automatically or manually. They are applied to curved plate and curved profile objects.

Ship direction

These marking lines can be created automatically or manually. They are applied to flat plate, curved plate, flat profile, curved profile, template, jig and remarking objects.

Labels

All labels. These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, curved profile, and template objects.

Fitting - plate to plate - butt joint

Line across two co-planar plates. These marking lines can be created automatically with arguments or manually. They are applied to flat plate and curved plate objects.

Fitting - plate to plate - tee joint

Short fitting lines on both plates on the joint edge. These marking lines can be created automatically with arguments or manually. They are applied to flat plate and curved plate objects.

Fitting - profile to plate - penetration

A pair of lines on the plate indicating the top of the profile. These marking lines can be created automatically with arguments or manually. They are applied to flat plate and curved plate objects.

Fitting - profile to plate

Short fitting lines on both plates on the joint edge. These marking lines are created automatically with arguments. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Seam control

A pair of lines parallel to the seam in defined distance to check accuracy of a connection. These marking lines can be created automatically with arguments or manually. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Hole

Includes air and drain holes. These marking lines are created automatically with arguments. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Reference plane lines

Waterlines, frames, and buttock lines. These marking lines are created automatically with

arguments. They are applied to flat plate, curved plate, flat profile, curved profile, and template objects.

Roll lines

Roll lines for shell rolling. These marking lines can be created automatically with arguments or manually. They are applied to curved plate objects.

Web frames check lines

Used for checking the accuracy of web frames cut out of plates. These marking lines are created automatically with arguments. They are applied to curved plate objects.

Bending control lines

Lines for checking bending of the cylindrical or conical shapes, such as hawse pipe. These marking lines are created automatically with arguments. They are applied to curved plate and curved profile objects.

Inverse bending line

These marking lines are created automatically with arguments. They are applied to curved profile objects.

Base control line

Marked on the template and shell plate. These marking lines are created automatically with arguments. They are applied to curved plate and template objects.

Site line

Marked on the template. These marking lines are created automatically with arguments. They are applied to template objects.

Plate edge check lines

Used for checking the accuracy of plates in fabrication and assemblies. These marking lines are created automatically with arguments. They are applied to flat plate and curved plate objects.

Block control

Reference line for all block panels. These marking lines are created manually. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Painting line

Only for shell plate. Border between different colors. These marking lines are created manually. They are applied to flat plate and curved plate objects.

Robot mark

Robot positioning mark. These marking lines are created manually. They are applied to flat plate and curved plate objects.

Mark (ship name)

Text that needs to be projected on a plate. These marking lines are created manually. They are applied to flat plate and curved plate objects.

User defined marking line

These marking lines are created manually. They are applied to flat plate, curved plate, flat profile, curved profile, template, and jig and remarking objects.

Margin

Assigned margin. These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Pin jig diagonal

Mapped to the individual part. These marking lines are created manually. They are applied to jig and remarking objects.

Diagonal

From the pin jig. In the case of a profile, it is a fitting mark on the profile. These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, curved profile, template, and jig and remarking objects.

Pin jig remarking additional

Mapped to the individual part. These marking lines are created manually. They are applied to jig and remarking objects.

Remarking additional

From the pin jig. In the case of a profile, it is a fitting mark on the profile. These marking lines are created automatically. They are applied to flat plate, curved plate, flat profile, and curved profile objects.

Marking Ribbon

Displays the controls used to create a marking line.



Properties

Activates the **Marking Line Properties** dialog box, which you use to view and modify the properties of the marking lines that you are about to place in the model. For more information, see *Marking Line Properties Dialog Box* (on page 84).



Part

Select the detailed plate or profile, or pin jig. You can select the part in a graphic view or in the **Workspace Explorer**.



Select Reference Curve

Select the reference curves that you want to use to create the marking line. This option is only available when the **Mode** is set to **Reference Curve**.



2D Sketch

Opens the Sketch 2D environment, where you sketch the marking line. This option displays when the **Mode** box is set to **2D Projection**.



Intersection

Select the intersection that you want to use to create the marking line. This option is only available when the **Mode** is set to **Intersection**.



Sketching Plane

Select the sketching plane for the marking line.

NOTE Depending upon the object you selected as the part, the sketching plane can be a

planar face from any of the following:

- Webs or flanges for profiles and members.
- Plate faces, when the base or offset is planar.
- Frames.



Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Draft 2D environment when you are drawing the marking.



Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Draft 2D environment when you are drawing the marking.



Sketch 2D

Activates the Sketch 2D environment in which you can draw the marking.



Reference Plane

Indicates that the marking line is at the intersection of the part and a reference plane. This option displays when the **Mode** box is set to **Intersection**.



Select Boundaries

Allows you to select marking line boundaries.



Select Related Part

Allows you to select detailed plate or profile parts to receive marking line attributes.

Finish

Creates the marking line that you have defined.



Cancel

Clears the selection.



Accept

Accepts the selection.



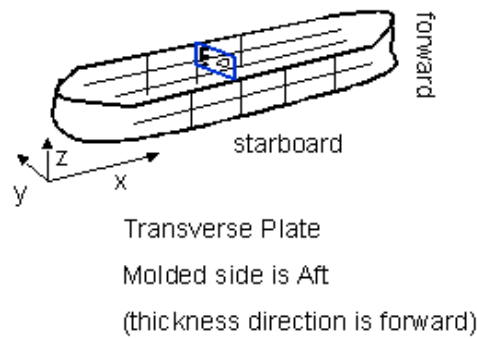
Solve Ambiguity

If the boundaries that you selected can form the object in more than one way, then you have defined an ambiguous solution. The software prompts you to select one or more solutions.

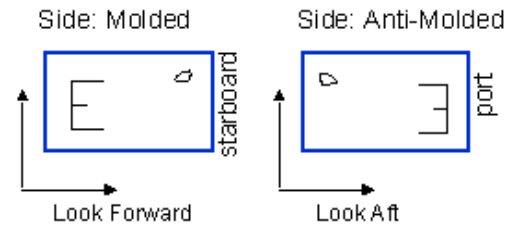
View

Specifies the orientation for the Sketch 2D view.

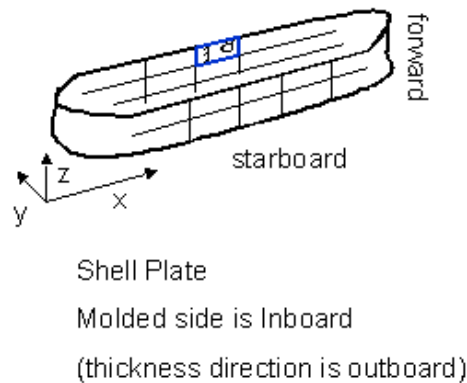
- **Standard** - The transverse view is looking forward. The longitudinal view is looking port. The plan view is looking down with forward to the right and port up.
- **Look at Side** - The view orientation is based on the **Side** property so that you are looking at the side that owns the mark.
- **Transverse Plate**



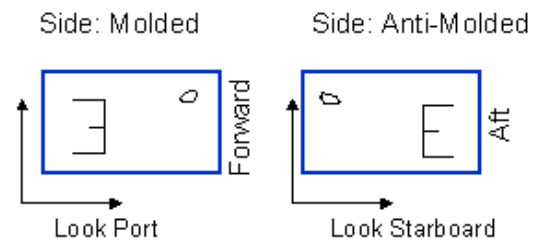
Sketch2D view orientations



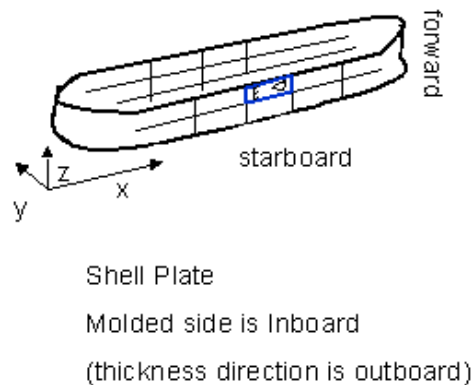
▪ Port Hull Plate



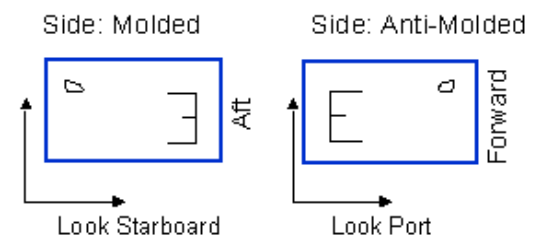
Sketch2D view orientations



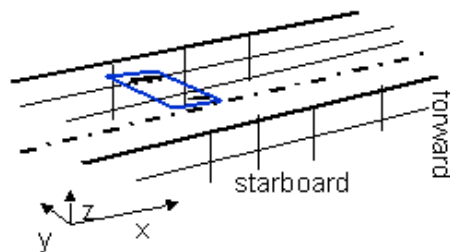
▪ Starboard Hull Plate



Sketch2D view orientations

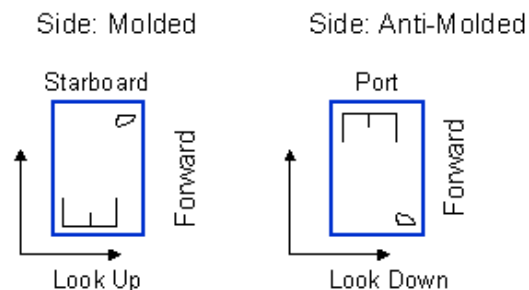


- **Deck Plate**



Deck Plate
Molded side is Below
(thickness direction is Above)

Sketch2D view orientations



Mode

Specifies whether the marking line is generated by two-dimensional projection, by intersection, or by reference curve.

- 2D Projection
- Intersection
- Reference Curve

Type

Select the type of marking line to place. The values available depend on the type of the selected parts.

Plate Mark Types:

- Block Mark
- Paint Mark
- Name Mark
- Robot Mark
- User Defined Mark
- Plate Location Mark
- Profile Location Mark
- Plate to Plate Tjoint Mark
- Profile to Plate Mark
- SeamControl Mark
- RollLine Mark
- RollBoundaryMark
- Knuckle Line Mark
- Naval Arch Ref Line

- ShipDirectionMark
- PinJigMarking
- Base Control Line Mark
- Bending Control Line Mark
- Bracket Location Mark
- Buttock Line Mark
- Collar Plate Loc Mark
- Diagonal Mark
- End Connection Mark
- FrameLine Mark
- Lap Connection Mark
- Plate Edge Check Mark
- TemplatePosMark
- WaterLine Mark
- Frame Check Lines Mark
- Mount Angle Mark
- Opening Mark
- Cutting Ref Mark
- Geometry Cut Mark
- End Cut Mark
- Hole Trace Mark
- Hole Ref Mark
- Seam Bend Mark
- Common Seam Mark
- Edge Location Mark
- Conn Part Mark
- Reference X Mark
- Reference Y Mark
- Reference Z Mark
- Reference Center Mark
- Lap Trace Mark
- Lap Ref Mark
- Lap Edge Mark
- Roll Section Ref Mark

Profile Mark Types:

- Block Mark
- Paint Mark
- Name Mark
- Robot Mark
- User Defined Mark
- End Connection Mark
- SeamControlMark
- FittingMark
- Plate Location Mark
- Knuckle Line Mark
- Mount Angle Mark
- Opening Mark
- Cutting Ref Mark
- Geometry Cut Mark
- End Cut Mark
- Hole Trace Mark
- Hole Ref Mark
- Profile to Profile Penetration
- Edge Location Mark
- Conn Part Mark
- Ship Direction Mark
- Lap Connection Mark
- WaterLine Mark
- Frameline Mark
- Buttock Line Mark
- Builtup Connection Mark


Member Mark Types:

- Block Mark
- Paint Mark
- Name Mark
- Robot Mark
- User Defined Mark

Pin Jig Mark Types:

- Extend PinJig Intersection
- PinJig Marking
- PinJig Diagonal
- Naval Arch Ref Line
- Mount Angle Mark
- Bending Interval Mark
- Annotation Mark
- Jig Floor Mark
- Opening Mark
- Cutting Ref Mark
- Geometry Cut Mark
- End Cut mark
- Hole Trace Mark
- Hole Ref Mark
- Feature Label Mark
- Profile to Profile Penetration
- Seam Bend Mark
- Common Seam Mark
- Edge Location Mark
- Conn Part Mark
- Knuckle Tab Mark
- Tab Ref Mark
- Reference X Mark
- Reference Y Mark
- Reference Z Mark
- Reference Center Mark
- Part Checklines Mark
- Lap Trace Mark
- Edge Fit Mark
- Lap Ref Mark
- Lap Edge Mark
- Knuckle Fit Mark
- Roll Section Mark
- Support Mark

- Ladder Mark
- Foundation Mark
- Stair Mark
- Paper Template Mark
- Builtup Connection Mark

 **NOTE** Do not create a marking line that overlaps the profile's remarking line. Extend the profile's marking line to meet the contour.

In this case, the pin jig output contains a marking line that takes the place of the original profile's marking line and the extension created in the previous steps.

This extended marking line can participate in intersection point creation, or not, depending on the custom rules for the type of the marking line that you extended.

Side

Select the side on which to define the marking line. The values available depend on the part type. The software displays an arrow indicating the side of the part on which the marking is placed.

- The options for plates include molded, anti-molded, base side, and offset side.
- The options for stiffeners include web left, web right, and top flange.
- The options for members include web left, web right, top flange, bottom flange, top flange right bottom, top flange left bottom, bottom flange right top, and bottom flange left top.
- The options for pin jigs include supported and remarking sides.

NOTES

- **MostMarking** - Side with the most marking lines. For the initial manual marking line, this is the side with the most automated marking (such as intersection marks).
- **MoldedSide** - Molded side.
- **Base Side** - The bottom side for decks based on the ship direction.
- **Offset Side** - The top side for decks based on the ship direction.
- **AssemblyOrientation** - Side based on assembly orientation (if the part is in an assembly).
- **AntiMoldedSide** - Anti-molded side (the thickness side).
- **MostStiffener** - Side on which the most stiffeners are mounted.
- **Web left** - Available for profiles.
- **Web right** - Available for profiles.
- **Top flange** - Available for profiles.

Define Plane Controls

Coincident Plane

Specifies that you want the plate part placed on the reference plane.

Offset from a Plane

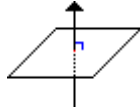
Places the plate system a specified distance from the reference plane. If you choose this option, you must define the offset distance.

Angle from plane

Places the plate part at a specified angle or slope to the reference plane. If you choose this option, you must define an axis of rotation and the angle or slope.

Plane by Point and Vector

Specifies the reference plane by defining a vector perpendicular to the reference plane. A third point defines the reference plane position along the vector.



Plane by Three Points

Specifies the reference plane using three points that you specify in the model.

Plane by Vectors Normal

Specifies the reference plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.

Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves. The **Lock** option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance for the plate part from the selected reference plane. You can specify the offset dynamically in graphics or by typing the distance. The **Offset** option is only available when you use the **Offset from a Plane** option.

Angle

Specify the angle at which to place the plate part relative to the reference plane. You have to define the axis of rotation using two points before you can define the angle. The **Angle** option is only available when you use the **Angle from plane** option.

Step

Specifies the angle step. The step is incremented by this value when the cursor is moved in the graphic view. The **Step** option is only available when you use the **Angle from plane** option.

Select Vector

Select or define a vector normal to the needed plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point

Specify the point along the vector at which to place the plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point 1

Specify the location of the first of three points that defines the plane. This option is only

available when you are using the **Plane by Three Points** option.

Define Point 2

Specify the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 3

Specify the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Select Plane

Select the plane to which your reference plane is to be normal. This option is only available when you are using the **Plane by Vectors Normal** option.

Select First Point

Specify the location of the first vector point. Your reference plane is parallel to this vector. This option is only available when you are using the **Plane by Vectors Normal** option.

Select Second Point

Specify the location of the second vector point. Your reference plane is parallel to this vector. This option is only available when you are using the **Plane by Vectors Normal** option.



Display Side

Indicates whether the arrow indicating the marking side displays.

What do you want to do?





- *Create a marking line by 2D projection* (on page [77](#))
 - *Create a marking line by intersection* (on page 79)
 - *Create a marking line by reference curve* (on page 80)
 - *Copy and paste a marking line* (on page 82)
 - *Modify marking* (on page 83)
 - *Delete marking* (on page 83)
-

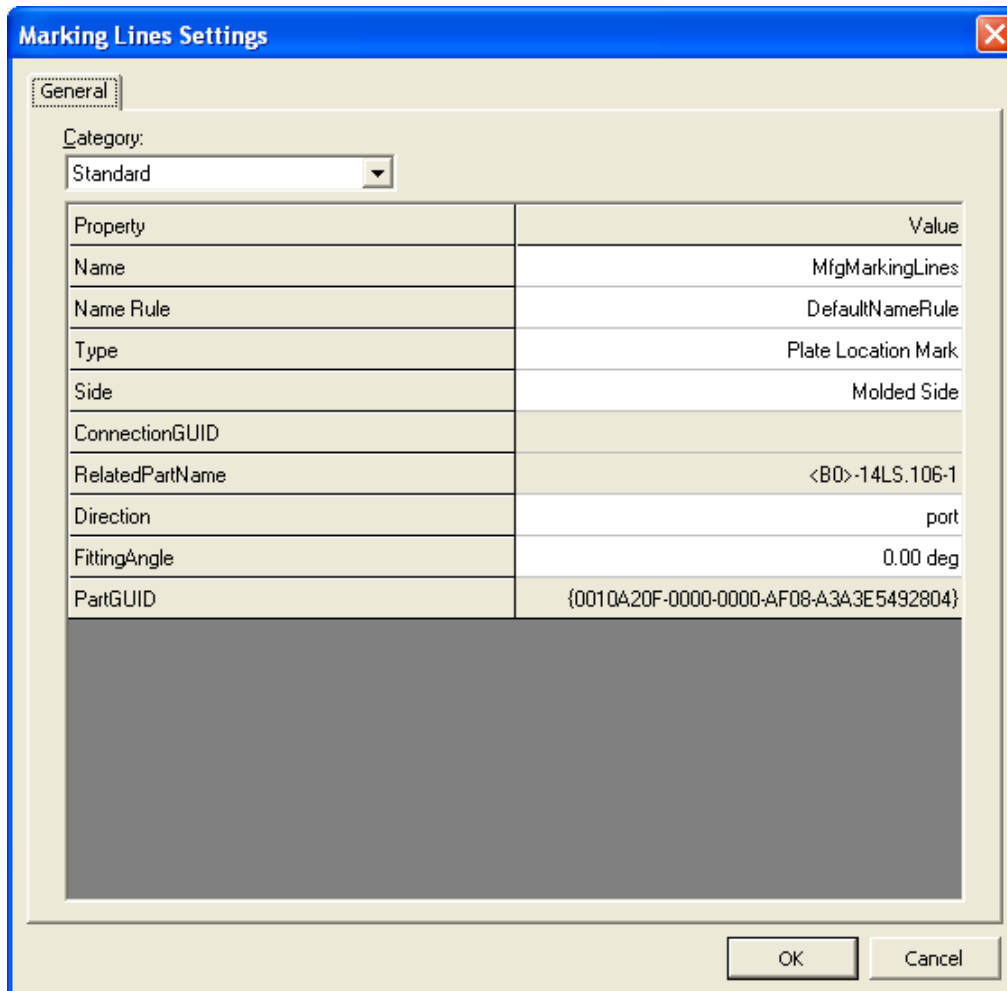
Create a marking line by 2D projection

1. Click **Marking** .
2. Select **2D Projection** from the **Mode** list.
3. On the **Marking** ribbon, click **Part** .
4. Select the part to mark.

TIPS

- You can select the parts in the model, or from the **Workspace Explorer**.

- A red arrow indicates the side of the part on which the markings are placed. Select a different value from the **Side** list to place the markings on a different side of the part.
- 5. Click **Sketching Plane** , and specify a sketching plane.
- 6. Click **2D Sketch** , and sketch the marking line.
- 7. Click **Finish** on the ribbon in the **2D Sketch** environment.
- 8. Click **Select Related Part** , and select the related parts to receive the marking line attributes.
- 💡 **TIP** You can select the parts in the model, or from the **Workspace Explorer**.
- 9. Click **Properties**  to review the marking attributes.
- 💡 **TIP** The attributes that display depend on the type of marking after the selection of the related part. For example, the **Connection GUID**, **Related Part Name**, **Direction**, and so on may display.





The image shows the 'Marking Lines Settings' dialog box with the 'General' tab selected. It contains a 'Category' dropdown set to 'Standard' and a table of properties and values. Below the table is a large grey rectangular area, and at the bottom are 'OK' and 'Cancel' buttons.

Property	Value
Name	MfgMarkingLines
Name Rule	DefaultNameRule
Type	Plate Location Mark
Side	Molded Side
ConnectionGUID	
RelatedPartName	<B0>-14LS.106-1
Direction	port
FittingAngle	0.00 deg
PartGUID	{0010A20F-0000-0000-AF08-A3A3E5492804}




- 10. Click **Finish**.

Create a marking line by intersection



1. Click **Marking** .
2. Select **Intersection** from the **Mode** list.
3. On the **Marking** ribbon, click **Part** .
4. Select the part to mark.


TIPS


- You can select the parts in the model, or from the **Workspace Explorer**.
- A red arrow indicates the side of the part on which the markings are placed. Select a different value from the **Side** list to place the markings on a different side of the part.

5. Click **Sketching Plane** , and specify a sketching plane.
6. Click **Reference Plane** , and specify a reference plane that intersects the plate.
7. Click **Select Boundaries** , and select the boundaries to define the marking line. For more information, see *Boundary Methods* (on page 43).

TIPS

- You can select the reference planes in the model, or from the **Workspace Explorer**.
 - You can select other markings on the part in the model as boundaries.
8. If the boundary definition results in more than one solution, click **Solve Ambiguity**  to choose the solution to use. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 45).
 9. Click **Select Related Part** , and select the related parts to receive the marking line attributes.

 **TIP** You can select the parts in the model, or from the **Workspace Explorer**.

10. Click **Properties**  to review the marking attributes.

TIP The attributes that display depend on the type of marking after the selection of the related part. For example, the **Connection GUID**, **Related Part Name**, **Direction**, and so on may display.

Marking Lines Settings

General



Category: Standard

Property	Value
Name	MfgMarkingLines
Name Rule	DefaultNameRule
Type	Plate Location Mark
Side	Molded Side
ConnectionGUID	
RelatedPartName	<B0>-14LS.106-1
Direction	port
FittingAngle	0.00 deg
PartGUID	{0010A20F-0000-0000-AF08-A3A3E5492804}

OK Cancel

11. Click **Finish**.

Create a marking line by reference curve


1. Click **Marking** .
2. Select **Reference Curve** from the **Mode** list.
3. On the **Marking** ribbon, click **Part** .
4. Select the part to mark.

TIPS



- You can select the parts in the model, or from the **Workspace Explorer**.
- A red arrow indicates the side of the part on which the markings are placed. Select a different value from the **Side** list to place the markings on a different side of the part.


5. Select the marking type to use from the **Type** list.
6. Select the reference curve to define the geometry for the marking line.


 **TIPS**

- You can select reference curves and logical connections in the model, or from the **System** tab of the **Workspace Explorer**.
 - You can select edge ports of the parts in the model.
7. Click **Select Boundaries** , and select the boundaries to define the marking line. For more information, see *Boundary Methods* (on page 43).

 **TIPS**

- You can select the reference planes in the model, or from the **Workspace Explorer**.
 - You can select other markings on the part in the model as boundaries.
8. If the boundary definition results in more than one solution, click **Solve Ambiguity**  to choose the solution to use. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 45).
 9. Click **Select Related Part** , and select the related parts to receive the marking line attributes.

 **TIP** You can select the parts in the model, or from the **Workspace Explorer**.

10. Click **Properties**  to review the marking attributes.

TIP The attributes that display depend on the type of marking after the selection of the related part. For example, the **Connection GUID**, **Related Part Name**, **Direction**, and so on may display.

Marking Lines Settings

General


Category: Standard

Property	Value
Name	MfgMarkingLines
Name Rule	DefaultNameRule
Type	Plate Location Mark
Side	Molded Side
ConnectionGUID	
RelatedPartName	<B0>-14LS.106-1
Direction	port
FittingAngle	0.00 deg
PartGUID	{0010A20F-0000-0000-AF08-A3A3E5492804}

OK Cancel

11. Click **Finish**.

Copy and paste a marking line

1. Click **Select**  on the vertical toolbar.
2. Select the marking lines to copy.

TIPS

- You can select the marking lines in the model, or from the **Workspace Explorer**.
- You can set the **Locate Filter** to **Marking** to make the selection easier.



3. Click **Edit > Copy**.
4. Click **Edit > Paste**.

5. Select the part on which to paste the marking lines.

TIPS



- You must select a detailed plate or profile for the destination.
 - If the marking that you copied is from a plate, you must select a plate for the destination. If the marking that you copied is from a profile, you must select a profile for the destination.
6. Select the sketching plane for the marking lines.
 7. Click **OK** on the **Paste** dialog box.

Modify marking

1. Click **Select**  on the vertical toolbar.
 2. Select the marking to modify.
-  **TIP** You can select the marking in the model, or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the marking as needed.

 **NOTE** You cannot change the parent plate when you modify the object.

Delete marking

1. Click **Select**  on the vertical toolbar.
2. Select the marking to delete.
3. Click **Delete** .

Marking Folder Properties Dialog Box

Specifies the properties for the marking folder that you are editing.

See Also

General Tab (Marking Folder Properties Dialog Box) (on page 83)

Relationship Tab (on page 86)

General Tab (Marking Folder Properties Dialog Box)

Name

Specifies the name of the marking folder. If you change the name, then the name rule changes automatically to **User Defined**.

Name Rule

Select a name rule for the marking folder.

Detailed Part Status

Displays the status of the parent object for the marking folder. This property only displays when you are modifying a marking folder.

Manufacturing Object Status

Displays the status of the marking folder. This property only displays when you are modifying a marking folder.

Marking Line Properties Dialog Box

Specifies the properties for the marking lines that you are editing.

See Also

General Tab (Marking Line Properties Dialog Box) (on page 84)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Marking Line Properties Dialog Box)

Name

Specifies the name of the marking line set. If you change the name, then the name rule changes automatically to **User Defined**.

Name Rule

Select a name rule for the marking line set.

- Select **DefaultNameRule** to use the following syntax: <"MK">_<Parent Name>-<Workshare Location ID>. For example, MK_<B0.1-7.1-4.1-2>-SH.1-1-1, where MK denotes a marking object, <B0.1-7.1-4.1-2>-SH.1-1 is the parent name, and 1 is the Workshare Location ID. If there is no parent, the parent name is "NoParent."
- Select **User Defined** to type a name in the **Name** box.

Type

Select the type of marking line to place.

Side

Select the side on which to place the marking line.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools > Recompute Objects**. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Physical Connection GUID

Displays the GUID of the physical connection. This property only displays when you have a specific type of marking line (example: Location Mark) and when the related part is selected.

Related Part Name

Displays the name of the related part. This property only displays when you have a specific type of marking line and when the related part is selected.

Direction

Select the direction of the related part. This property only displays when you have a specific type of marking line and when the related part is selected.

Fitting Angle

Specifies the fitting angle. This property only displays when you have a specific type of marking line and when the related part is selected.

Part GUID

Displays the GUID of the related part. This property only displays when you have a specific type of marking line and when the related part is selected.

Flange Direction

Specifies the flange direction. This property only displays when you have a specific type of marking line and when the related part is selected.

Max Assembly Margin Value

Specifies the maximum assembly margin value. This property only displays when you have a specific type of marking line and when the related part is selected.

Max Fabrication Margin Value

Specifies the maximum fabrication margin value. This property only displays when you have a specific type of marking line and when the related part is selected.

Max Custom Margin Value

Specifies the maximum custom margin value. This property only displays when you have a specific type of marking line and when the related part is selected.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

 **NOTE** You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.


Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer


Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or

project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

 **NOTE** The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

 **NOTE** You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.


New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group


Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write

access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

 **NOTE** We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

 **NOTE** Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.


Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Marking by Definition


 Creates marking lines based on symbol definition.

Marking by Definition Dialog Box (on page 90)


What do you want to do?

- *Create a skid mark* (on page 89)
 - *Create a bilge keel mark* (on page 89)
 - *Create a block mark* (on page 90)
-


Create a skid mark

1. Click  **Marking by Definition** on the vertical toolbar.
*The **Marking by Definition** dialog box displays.*
2. In the **Select marking line definition** box, select **Skid Mark**.
3. Set the remaining parameters to meet your needs. You may need to click **Expand** to display the offset parameters.

Create a bilge keel mark

1. Click  **Marking by Definition** on the vertical toolbar.
*The **Marking by Definition** dialog box displays.*
2. In the **Select marking line definition** box, select **Bilge Keel Mark**.
3. Set the remaining parameters to meet your needs.

Create a block mark

1. Click  **Marking by Definition** on the vertical toolbar.
*The **Marking by Definition** dialog box displays.*
2. In the **Select marking line definition** box, select **Block Mark**.
3. Set the remaining parameters to meet your needs.

Marking by Definition Dialog Box

Specifies the properties for the marking lines.

Select marking line definition

Select the type of marking line definition from the list. This specifies the *marking by definition* symbol defined in the catalog. The marking line definition determines the parameters that display on the dialog box.

Preview

Displays a preview of the marking lines.

Bilge Keel Mark

Plate System

Select the plate system associated with the marking line.

Plate/Profile System

Select the connected plate or profile system associated with the marking line.

Reference Surface

Select the reference surface of connected system parts associated with the marking line.

Marking Side

Select the side on which the marking lines are placed.

Marking Type

Select the marking type for the marking lines.

Name Rule

Select the naming rule for the marking lines.

Apply

Creates the marking line using the parameters specified on the dialog box. The parameters remain active on the dialog box.

OK

Creates the marking line using the parameters specified on the dialog box. The parameters are cleared from the dialog box.

Skid Mark

Plate System

Select the plate system associated with the marking line.

Lower Frame

Select the lower frame associated with the marking line.

Upper Frame

Select the upper frame associated with the marking line.

Longitudinal Reference

Select the longitudinal reference associated with the marking line.

Plane Reference

Select the plane reference associated with the marking line.

Marking Type

Select the marking type for the marking lines.

Marking Side

Select the side on which the marking lines are placed.

Name Rule

Select the naming rule for the marking lines.

Expand

Expands the dialog box to display the offset parameters for the marking line.

Starting Plane

Specify the starting plane from which the skid mark is created. The skid mark is placed from the starting plane to the next frame. This option displays only if you click the **Expand** button.

Offset

Specify the girth offset distance from the longitudinal reference. This option displays only if you click the **Expand** button.

Add

Adds a new row to the table. This option displays only if you click the **Expand** button.

Delete

Deletes the active row. If no row is active, deletes the first row in the table. This option displays only if you click the **Expand** button.

Block Mark

Pin Jig

Specifies the pin jig associated with the assembly.

Offset Value

Specifies the offset distance from the pin jig base plane where marks are created on plates connected to plates supported by the pin jig.

Supported Plates by Type

Specifies the type of supported plate. The available options include:

- **Transversal** - Software automatically selects all of the transversal plates that are connected to the plates supported by the pin jig.
- **Longitudinal** - Software automatically selects all of the longitudinal plates that are connected to the plates supported by the pin jig.
- **Transversal and Longitudinal** - Software automatically selects both the transversal and the longitudinal plates that are connected to the plates supported by the pin jig.
- **User Selection** - Nothing is selected automatically.

Supported Plates by User

Select individual plates that are connected to plates supported by the pin jig. This option is only available if **Supported Plates by Type** is set to **User Selection**.

Marking Type

Specifies the marking type for the marking lines.

Marking Side

Specifies the side on which the marking lines are placed.

Name Rule

Specifies the naming rule for the marking lines.

SECTION 3


Margins

The margin commands add material at plate edges or profile ends.

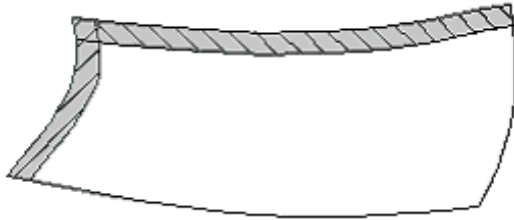
In This Section

Fabrication Margin	93
Connect Fabrication Margin	105
Margin By Assembly Connections	115
Regional Margin.....	122

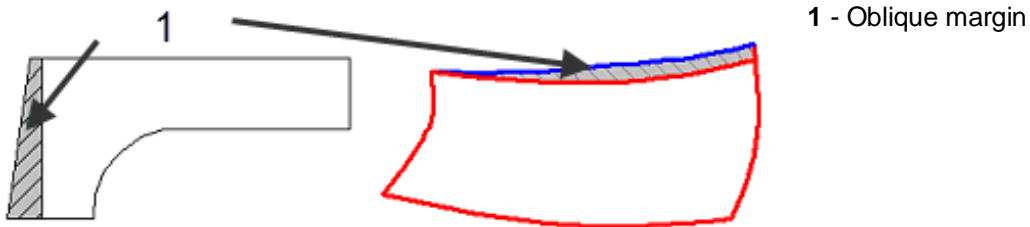
Fabrication Margin

 Adds material at plate edges or profile ends. The material is removed after bending or assembly if it is no longer needed.

You can place simple constant value margins.



You can also place oblique (variable) margins. For an oblique margin, the value is linearly distributed between the start and end points. Oblique margins are only applied to plates.



The following fabrication margin types are delivered with the software:

Bending margin

Extra material applied to the manufacturing part to ensure proper object shape and size after bending.

Bending margin is a plate or profile part before bending. The margin is used during the bending work on the part using a bending machine. The bending margin is applied parallel to the edge of the part. You can apply bending margins to all sides of the part. The bending margin is removed after the bending work of the part is completed. The workflow in the shop is as follows:

- Cut the part including the bending margin.
- Bend the curved part.
- After the bending work is complete, remove any remaining bending margin.

Heating margin

Extra material applied to the manufacturing part to ensure proper object shape and size after heating.

Heating margin is a margin applied to a part that is bent by heating, or to a part that is known to distort during the manufacturing process and requires heating to remove the distortion. The margin is used by the heating process. Heating margins are applied as oblique or parallel margins to the edge of the part. The workflow in the shop is as follows:

- Cut the part including the heating margin.
- Heat the part.

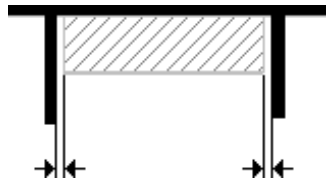
End face margin

Extra material applied to the manufacturing part that is consumed as part of the assembly process. Extra material is typically added parallel to the part end face or edge at the assembly or block lateral face.

During assembly, the margin gradually shrinks and eventually disappears after completion of the assembly or block.

Intercostal margin

Material that is *removed* at the ends of a manufacturing part when the ends are fillet welded.



Generic Margin - Non Trimmed

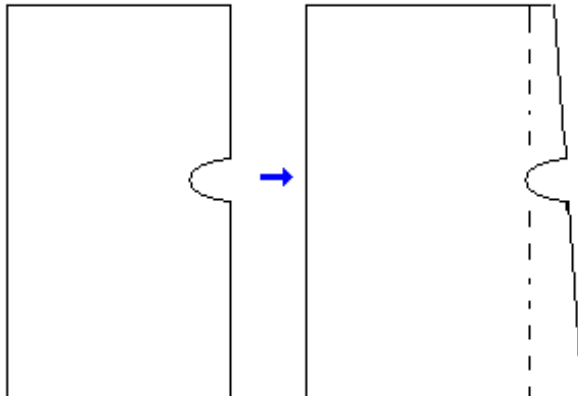
Extra material applied to the manufacturing part that is consumed during manufacturing.

Generic Margin - Trimmed

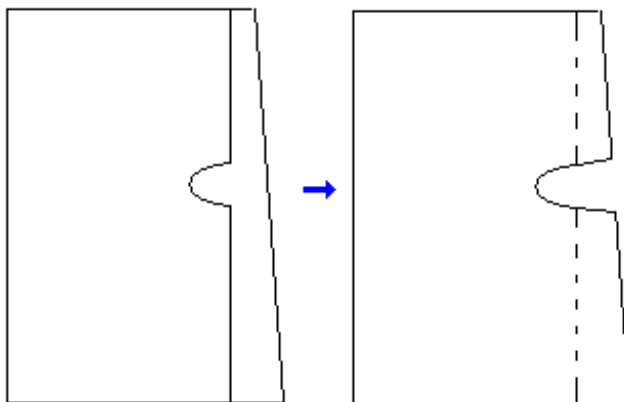
Extra material applied to the manufacturing part that should be removed after manufacturing.

If the edge of the plate contains features, you can specify that those features remain fixed, or move with the margin.

The following example shows an edge feature that moves with the fabrication margin:

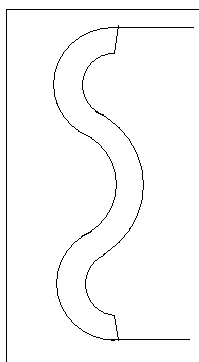


The following example shows an edge feature that does not move with the fabrication margin:

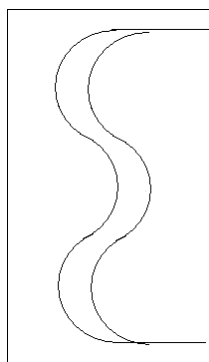


NOTE For built-up profiles, you can define the margin for the web and flange separately.

Margins can be applied in two different ways, depending upon the rules. Margins can be applied by translation, or by applying a constant offset through the length of the curve.



Offset



Translated

Coaming

Add the additional material necessary to account for your modeling practice.

PanelEnd

Adds material to support the 2-stage production cutting process of plates. Smart 3D applies the solution in combination with panel assembly and panel welding sequence.

By default, the margin is applied by translation for all profiles and members. That is, the contour curve is shifted in the margin direction. To apply a margin by constant offset, you must modify the rules. For more information, see the *Structural Manufacturing Reference Data Guide*.

Fabrication Margin Ribbon

Displays the controls used to add a fabrication margin to plate edges or profile ends.

Properties

Opens the **Fabrication Margin Properties** dialog box, which you use to view and modify the properties of the fabrication margin that you are about to place in the model. For more information, see *Fabrication Margin Properties Dialog Box* (on page 102).

Part

Select the detailed plate, profile, or member part to which you want to add the fabrication margin. You can select the part in a graphic view or in the **Workspace Explorer**.

NOTES

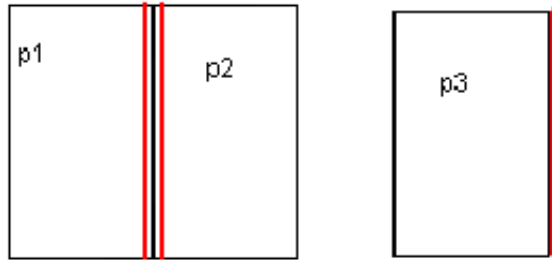
- If the **Rule** box is selected and you first select a plate that is oblique by rule, then the oblique properties are set and you can pick only connected plates.
- If the **Rule** box is selected and you first select a plate that is constant by rule, then you can pick only plates that comply with that rule.
- If the **Rule** box is not selected, then you can select any plates. The software applies the last used margin as a default.
- If the **Mode** box is set to **Constant**, then you can select any parts.
- If the **Mode** box is set to **Oblique**, then you can select only connected parts.
- If you clear a part, then the software automatically clears the faces of that part.

Lateral Face

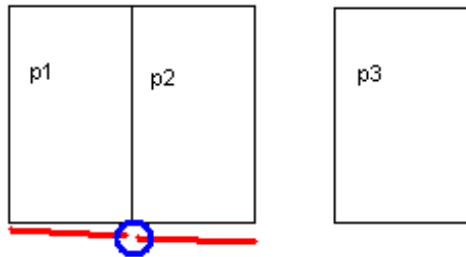
Select the plate later faces (edges) or profile ends on which to add margin. Because a face can only have one margin of a given type, the faces of the selected parts that already have a margin of the currently selected type are highlighted and not selectable.

NOTES

- If the **Mode** box is set to **Constant**, then you can select any face of the selected parts. If you select two parts, and then select the face between those parts, the software selects one edge of the connected parts. You must select an edge on the other plate for placing multiple margins. You can place a margin on only one edge of the plat. For example, if you select p1, p2, and p3 in the **Part** step, then you can select one edge on each of the plates to place a margin.



- If the **Mode** box is set to **Oblique**, then you can select only connected, continuous faces. For example if you select p1 and p2, then you cannot select p3.



- After the faces are selected, then the software runs the appropriate rules to set the margins and the default feature move setting.

Start End

Select the start point for the oblique margin on one side of the selected faces. This option is only available if the **Mode** box is set to **Oblique**.

Finish

Creates the fabrication margin using the parameters that you have defined.

Cancel

Clears the selection.

Accept

Accepts the selection.

Part Filter

Specifies the type of part that you can select.

Rule

Specifies that the margin is placed by rule.

Mode

Specifies whether the margin is constant or oblique.

NOTE If you have selected multiple plates or faces, you cannot change from constant to oblique. You can change multiple plates or faces from oblique to constant.

Type

Select the type of margin to apply. For more information on the available types, see *Fabrication Margin* (on page 93).

Value

Specifies the amount of margin to apply. This option displays when The **Mode** box is set to **Constant**.

Start Value

Specifies the amount of margin to apply at the identified start point of the selected lateral face. This option displays when The **Mode** box is set to **Oblique**.

End Value

Specifies the amount of margin to apply at the opposite point of the identified start point at the selected lateral face. The oblique margin is linearly distributed between the start and end points. This option displays when the **Mode** box is set to **Oblique**.

Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.

Geometry Change

Indicates whether the margin should be trimmed. The following options are available:

- **As Margin** - Trimming is needed. The extension is only on the edge.
- **By Stretch** - No trimming is needed. The software stretches the shape along the part like shrinkage.
- **As Reference** - No request to extend the geometry. The margin is placed as a reference.





Stretch Offset

The length of the adjacent edge that the software adjusts while applying the margin. This option is only editable if **Geometry Change** is set to **By Stretch**.

What do you want to do?

- *Add fabrication margin* (on page 99)
- *Modify fabrication margin* (on page 100)
- *Copy and paste a constant fabrication margin* (on page 101)
- *Copy and paste an oblique fabrication margin* (on page 101)
- *Delete fabrication margin* (on page 102)

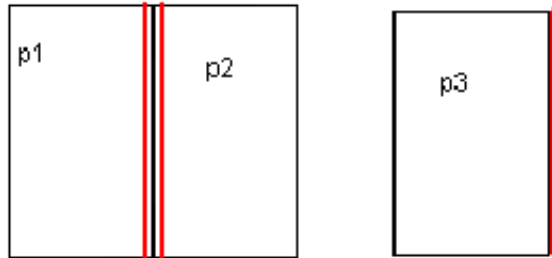
Add fabrication margin

1. Click **Fabrication Margin** .
2. On the **Fabrication Margin** ribbon, click **Part** .
3. Select the parts for which you want to add a fabrication margin.
TIP You can select the plate parts in the model, or from the **Workspace Explorer**.
4. Click **Lateral Face** .
5. Select the plate lateral faces (edges) or profile ends on which to add the margin.
6. Select the margin mode.
7. Select the margin type.
8. Type the margin values.
9. Select the margin direction.
10. Specify the margin geometry change and stretch offset values.
11. Click **Accept** .
12. Click **Finish**.

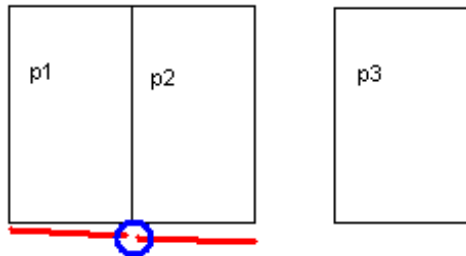
NOTES

- If the **Rule** box is selected and you first select a plate that is oblique by rule, then the oblique properties are set and you can pick only connected plates.
- If the **Rule** box is selected and you first select a plate that is constant by rule, then you can pick only plates that comply with that rule.
- If the **Rule** box is not selected, then you can select any plates. The software applies the last used margin as a default.
- If the **Mode** box is set to **Constant**, then you can select any parts.
- If the **Mode** box is set to **Oblique**, then you can select only connected parts.
- If you deselect a part, then the software automatically deselects the faces of that part.

- If the **Mode** box is set to **Constant**, then you can select any face of the selected parts. If you select two parts, and then select the face between those parts, the software selects one edge of the connected parts. You must select an edge on the other plate for placing multiple margins. You can place a margin on only one edge of the plate. For example, if you select p1, p2, and p3 in the **Part** step, then you can select one edge on each of the plates to place a margin.




- If the **Mode** box is set to **Oblique**, then you can select only connected, continuous faces. For example if you select p1 and p2, then you cannot select p3.




- After the faces are selected, then the software runs the appropriate rules to set the margins and the default feature move setting.

Modify fabrication margin

1. Click **Select**  on the vertical toolbar.
2. Select the margin to modify.
 - 💡 **TIP** You can select the plate parts in the model, or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the margin as needed.
 - 📌 **NOTE** You cannot change the parent plate when you modify the object.

Copy and paste a constant fabrication margin

1. Click **Select**  on the vertical toolbar.
2. Select the margin to copy.


TIPS

- You can select the margin in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Constant Margin** to make the selection easier.
3. Click **Edit > Copy**.
 4. Click **Edit > Paste**.
 5. Select the plate lateral faces (edges) or profile ends on which to paste the margin.

TIPS

- You must select a destination that does not already have a margin of the type that you are pasting. That is, a plate or profile cannot have duplicate margins of the same type.
 - If the margin that you copied is from a plate, you must select a plate for the destination. If the margin that you copied is from a profile, you must select a profile for the destination. If the margin that you copied is from a member, you must select a member for the destination.
6. Click **OK** on the **Paste** dialog box.

Copy and paste an oblique fabrication margin

1. Click **Select**  on the vertical toolbar.
2. Select the margin to copy.



TIPS

- You can select the margin in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Oblique Margin** to make the selection easier.
3. Click **Edit > Copy**.
 4. Click **Edit > Paste**.
 5. Select the plate lateral faces (edges) or profile ends on which to paste the margin.

TIPS

- You must select a destination that does not already have a margin of the type that you are pasting. That is, a plate or profile cannot have duplicate margins of the same type.
 - If the margin that you copied is from a plate, you must select a plate for the destination. If the margin that you copied is from a profile, you must select a profile for the destination. If the margin that you copied is from a member, you must select a member for the destination.
6. Click **OK** on the **Paste** dialog box.

Delete fabrication margin

1. Click **Select**  on the vertical toolbar.
2. Select the fabrication margin to delete.
3. Click **Delete** .

Fabrication Margin Properties Dialog Box

Specifies the properties for the fabrication margin that you are editing.

See Also

General Tab (Fabrication Margin Properties Dialog Box) (on page 102)

Feature Tab (Fabrication Margin Properties Dialog Box) (on page 105)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Fabrication Margin Properties Dialog Box)

Name

Specifies the name of the part on which the margin is being created or modified. This column is not editable.

Status

Indicates whether the margin is being created or modified. This column is not editable.

Rule

Indicates whether the margin properties come from the rule or are user defined. The possible values are **Apply** and **Ignore**. By default, this value is set to **Apply** for all margins that are being created. If you change the values, the rule status automatically changes to **Ignore**. For the existing margins, the status is **Ignore**. This column is editable.

Mode

Indicates the mode of the margin object. This column is not editable.

Type

Type of margin to apply. This column is editable.

Bending margin

Extra material applied to the manufacturing part to ensure proper object shape and size after bending.

Bending margin is a plate or profile part before bending. The margin is used during the bending work on the part using a bending machine. The bending margin is applied parallel to the edge of the part. You can apply bending margins to all sides of the part. The bending margin is removed after the bending work of the part is completed. The workflow in the shop is as follows:

- Cut the part including the bending margin.
- Bend the curved part.

- After the bending work is complete, remove any remaining bending margin.

Heating margin

Extra material applied to the manufacturing part to ensure proper object shape and size after heating.

Heating margin is a margin applied to a part that is bent by heating, or to a part that is known to distort during the manufacturing process and requires heating to remove the distortion. The margin is used by the heating process. Heating margins are applied as oblique or parallel margins to the edge of the part. The workflow in the shop is as follows:

- Cut the part including the heating margin.
- Heat the part.

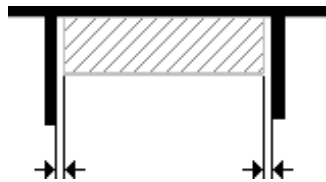
End face margin

Extra material applied to the manufacturing part that is consumed as part of the assembly process. Extra material is typically added parallel to the part end face or edge at the assembly or block lateral face.

During assembly, the margin gradually shrinks and eventually disappears after completion of the assembly or block.

Intercostal margin

Material that is *removed* at the ends of a manufacturing part when the ends are fillet welded.



Generic Margin - Non Trimmed

Extra material applied to the manufacturing part that is consumed during manufacturing.

Generic Margin - Trimmed

Extra material applied to the manufacturing part that should be removed after manufacturing.

Margin Value

Specifies the amount of margin to apply. This option displays when **Constant Margin** is selected in the **Mode** box.

Start Value

Specifies the amount of margin to apply at the identified start point of the selected lateral face. This option displays when **Oblique Margin** is selected in the **Mode** box.

End Value

Specifies the amount of margin to apply at the opposite point of the identified start point at the selected lateral face. The oblique margin is linearly distributed between the given start and end points. This option displays when **Oblique Margin** is selected in the **Mode** box.

Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the

direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.

Geometry Change

Indicates whether the margin should be trimmed. The following options are available:

As Margin - Trimming is needed. The extension is only on the edge.

By Stretch - No trimming is needed. The software stretches the shape along the part like shrinkage.

As Reference - No request to extend the geometry. The margin is placed as a reference.

Stretch Offset

The length of the adjacent edge that the software adjusts while applying the margin. This is only editable if the **Geometry Change** is set to **By Stretch**.

Action

Indicates whether or not the margin properties are applied to the margin object. The possible values for objects that you are creating are **Apply** and **Ignore**. The possible values for objects that you are modifying are **Apply**, **Ignore**, and **Delete**. If the value is set to **Apply**, the modified properties are applied to the margin object. If the value is set to **Ignore**, then the modified properties are not applied to the margin object. If the value is set to **Delete**, then the margin object is deleted.

Add Margin

Adds a new row to the grid. This new row indicates the properties of another margin object that are created at the same port of the selected part. By default, all of the properties of the selected margin are copied onto the new margin object except for the margin type. You can edit these properties.

Rule

Indicates whether or not margins are placed by rule.

Margin Type

Indicates the group margin type. This is the same value as shown in the ribbon bar.

Margin Value

Indicates the group margin value. This is the same value as shown in the ribbon bar.

Offset Direction

Indicates the offset direction. This is the same value as shown in the ribbon bar.

Update Dependent Objects

Indicates whether or not the dependent object parameters need to be updated if the parent margin properties change.

Feature Tab (Fabrication Margin Properties Dialog Box)

Feature name

Specifies the name of the feature. Features are listed in order from one end of the face to the other. When you select a feature on the dialog box, that feature highlights in the model.

Condition

Specifies the movement option for the feature when the margin is applied. Options include **Fix** or **Move**.


Set by Rule

Sets the movement condition for all selected features as specified by the rules.

See Also

Fabrication Margin Properties Dialog Box (on page 102)

Connect Fabrication Margin

 Selects all of the plates or profiles bounded to the plate system or part, and selects all the faces at the bounding plate systems or parts to place margins.

Parts can be bounded to both sides of the plate system. Use the toggle button to select parts connected to either side.

Connect Fabrication Margin Ribbon

Displays the controls used to add a fabrication margin to all of the plate edges or profile ends connected to a selected part.



Fabrication Margin Settings

Opens the **Fabrication Margin Settings** dialog box, which you use to view and modify the properties of the fabrication margin that you are about to place in the model. For more information, see *Margin Settings Dialog Box* (on page 109).



Connected Part

Select the bounding plate system or part. The software highlights all of the connected parts on one side of your selection and displays an arrow indicating the active side. Click the **Direction** toggle to highlight the connected parts on the other side of your selection.



Part

Select the plate or profile part to which you want to add the fabrication margin. You can select the part in a graphic view or in the **Workspace Explorer**.

NOTES

- Connected fabrication margins are always **Constant**.
- If the **Rule** box is not selected, then you can select any plates. The software applies the last used margin as a default.
- If you clear a part, then the software automatically clears the faces of that part.



Lateral Face

Select the plate later faces (edges) or profile ends on which to add margin. Because a face can only have one margin of a given type, the faces of the selected parts that already have a margin of the currently define type assigned are highlighted and not selectable.



Margin Properties

Displays a dialog box that allows you to review or edit the properties of the margins that you are creating or editing. For more information, see *Margin Properties Dialog Box (Connect Fabrication Margin)* (on page 112).

Finish

Creates the fabrication margin using the parameters that you have defined.



Cancel

Clears the selection.



Accept

Accepts the selection.

Name Rule

Specifies the naming rule for the margin objects that you are creating. The naming rule is not applied on margin objects that you are modifying.

Connected Part Filter

Specifies the filter for selecting connected parts. The available options are provided by the fabrication margin rule. By default, the following options are available:

- **Plate** - Allows you to select plate parts.
- **Profile** - Allows you to select profile parts.
- **Plate System** - Allows you to select plate systems.
- **Member** - Allows you to select members.



Direction

Toggles the side of the side of the bounding plate system or part that the software uses to find connected parts. A red arrow displays in the model to indicate the current direction.

Part Filter

Specifies the filter for selecting parts. The available options are provided by the fabrication margin rule. By default, the following options are available:

- **Plate** - Allows you to select plate parts.
- **Profile** - Allows you to select profile parts.
- **Member** - Allows you to select members.
- **Bracket** - Allows you to select brackets.
- **All** - Allows you to select plates, profiles, members, and brackets. This is a combination of all other filters. This option respects the **Profile with SetBack** option set on the **Properties** dialog box.
- **All with Setback** - Allows you to select plates, profiles (including profiles with setback), members, and brackets.

- **Customize** - Opens the **New** tab on the **Properties** dialog box.

Rule

Specifies that the margin is placed by rule.

Type

Select the type of margin to apply. For more information on the available types, see *Fabrication Margin* (on page 93).

Value

Specifies the amount of margin to apply. This option displays when The **Mode** box is set to **Constant**.

Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.

Geometry Change

Indicates whether the margin should be trimmed. The following options are available:

- **As Margin** - Trimming is needed. The extension is only on the edge.
- **By Stretch** - No trimming is needed. The software stretches the shape along the part like shrinkage.
- **As Reference** - No request to extend the geometry. The margin is placed as a reference.


Stretch Offset

The length of the adjacent edge that the software adjusts while applying the margin. This option is only editable if **Geometry Change** is set to **By Stretch**.


What do you want to do?

- *Add connected fabrication margin* (on page 108)
 - *Modify a connected fabrication margin* (on page 108)
 - *Delete a connected fabrication margin* (on page 108)
-

Add connected fabrication margin

1. Click **Connect Fabrication Margin** .
2. Select the bounding plate system or part based on the filter criteria that you have defined using the **Connected Part Filter** box.

TIPS

- You can select the plate parts in the model, or from the **Workspace Explorer**.
- The software highlights the connected parts on one side of the bounding object. Click the **Direction**  toggle to select the connected parts on the other side of the bounding object.

3. Click Fabrication **Margin Settings** .



*The **Margin Properties** dialog box displays.*

4. Set the margin settings to meet your needs. For more information, see *Margin Properties Dialog Box (Connect Fabrication Margin)* (on page 112).
5. Click **Finish**.



NOTES

- Connected fabrication margins are always **Constant**.
- If the **Rule** box is not selected, then you can select any plates. The software applies the last used margin as a default.
- If you clear a part, then the software automatically clears the faces of that part.

Modify a connected fabrication margin

1. Click **Select**  on the vertical toolbar.
2. Select the connected fabrication margin to modify.
 **TIP** You can select the assembly margins from the **Workspace Explorer**.
3. Using the ribbon controls, edit the margin as needed.

Delete a connected fabrication margin

1. Click **Select**  on the vertical toolbar.
2. Select the connected fabrication margin to delete.
3. Click **Delete** .

Margin Settings Dialog Box

Displays options for viewing or editing common properties of the selected margin objects.

- *General Tab (Margin Properties Dialog Box)* (on page 109)
- *Part Filters (Margin Properties Dialog Box)* (on page 112)

Name

Specifies the name of the assembly margin.

Naming Rule

Specifies the naming rule applied to the margin objects to create. Select **User Defined** to type a name in the **Name** box.

Margin Naming Rule

Specifies the naming rule for the assembly margin children.

Margin Type

Specifies the common margin type of the margin objects.

Margin Value

Specifies the common margin value of the margin objects.

Margin Direction

Specifies the common margin direction of the margin objects.

Geometry Change

Specifies the common margin geometry change of the margin objects.

Stretch Offset

Specifies the common margin stretch offset value of the margin objects.

General Tab (Margin Properties Dialog Box)

Part

Specifies the name of the part on which the margin is being created or modified. This column is not editable.

Status

Indicates whether the margin is being created or modified. This column is not editable.

Rule

Indicates whether the margin properties come from the rule or are user defined. The possible values are **Apply** and **Ignore**. By default, this value is set to **Apply** for all margins that are being created. If you change the values, the rule status automatically changes to **Ignore**. For the existing margins, the status is **Ignore**. This column is editable.

Mode

Indicates the mode of the margin object. This column is not editable.

Type

Type of margin to apply. This column is editable.

Bending margin

Extra material applied to the manufacturing part to ensure proper object shape and size after bending.

Bending margin is a plate or profile part before bending. The margin is used during the bending work on the part using a bending machine. The bending margin is applied parallel to the edge of the part. You can apply bending margins to all sides of the part. The bending margin is removed after the bending work of the part is completed. The workflow in the shop is as follows:

- Cut the part including the bending margin.
- Bend the curved part.
- After the bending work is complete, remove any remaining bending margin.

Heating margin

Extra material applied to the manufacturing part to ensure proper object shape and size after heating.

Heating margin is a margin applied to a part that is bent by heating, or to a part that is known to distort during the manufacturing process and requires heating to remove the distortion. The margin is used by the heating process. Heating margins are applied as oblique or parallel margins to the edge of the part. The workflow in the shop is as follows:

- Cut the part including the heating margin.
- Heat the part.

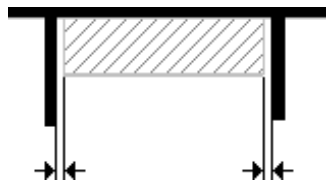
End face margin

Extra material applied to the manufacturing part that is consumed as part of the assembly process. Extra material is typically added parallel to the part end face or edge at the assembly or block lateral face.

During assembly, the margin gradually shrinks and eventually disappears after completion of the assembly or block.

Intercostal margin

Material that is *removed* at the ends of a manufacturing part when the ends are fillet welded.



Generic Margin - Non Trimmed

Extra material applied to the manufacturing part that is consumed during manufacturing.

Generic Margin - Trimmed

Extra material applied to the manufacturing part that should be removed after manufacturing.

Start Value

Specifies the amount of margin to apply at the identified start point of the selected lateral face. This option displays when **Oblique Margin** is selected in the **Mode** box.

End Value

Specifies the amount of margin to apply at the opposite point of the identified start point at the selected lateral face. The oblique margin is linearly distributed between the given start and end points. This option displays when **Oblique Margin** is selected in the **Mode** box.

Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.

Geometry Change

Indicates whether the margin should be trimmed. The following options are available:

- **As Margin** - Trimming is needed. The extension is only on the edge.
- **By Stretch** - No trimming is needed. The software stretches the shape along the part like shrinkage.
- **As Reference** - No request to extend the geometry. The margin is placed as a reference.

Stretch Offset

The length of the adjacent edge that the software adjusts while applying the margin. This is only editable if the **Geometry Change** is set to **By Stretch**.

Action

Indicates whether or not the margin properties are applied to the margin object. The possible values for objects that you are creating are **Apply** and **Ignore**. The possible values for objects that you are modifying are **Apply**, **Ignore**, and **Delete**. If the value is set to **Apply**, the modified properties are applied to the margin object. If the value is set to **Ignore**, then the modified properties are not applied to the margin object. If the value is set to **Delete**, then the margin object is deleted.

Rule

Indicates whether or not margins are placed by rule.

Type

Indicates the group margin type. This is the same value as shown in the ribbon bar.

Value

Indicates the group margin value. This is the same value as shown in the ribbon bar.

Update Dependent Objects

Indicates whether or not the dependent object parameters need to be updated if the parent margin properties change.

Add Margin

Adds a new row to the grid. This new row indicates the properties of another margin object that to create at the same port of the selected part. By default, all of the properties of the selected margin are copied onto the new margin object except for the margin type. You can edit these properties.

Part Filters (Margin Properties Dialog Box)**Plate**

Indicates whether you can select plate parts.

Profile

Indicates whether you can select profile parts.

Member

Indicates whether you can select member parts.

Bracket

Indicates whether you can select bracket parts.

Profile with Set Back

Indicates whether profiles with set back (with only logical connections) are considered for margins.

Margin Properties Dialog Box (Connect Fabrication Margin)**Part**

Specifies the name of the part on which the margin is being created or modified. This column is not editable.

Status

Indicates whether the margin is being created or modified. This column is not editable.

Rule

Indicates whether the margin properties come from the rule or are user defined. The possible values are **Apply** and **Ignore**. By default, this value is set to **Apply** for all margins that are being created. If you change the values, the rule status automatically changes to **Ignore**. For the existing margins, the status is **Ignore**. This column is editable.

Mode

Indicates the mode of the margin object. This column is not editable.

Type

Type of margin to apply. This column is editable.

Bending margin

Extra material applied to the manufacturing part to ensure proper object shape and size after bending.

Bending margin is a plate or profile part before bending. The margin is used during the bending work on the part using a bending machine. The bending margin is applied parallel to

the edge of the part. You can apply bending margins to all sides of the part. The bending margin is removed after the bending work of the part is completed. The workflow in the shop is as follows:

- Cut the part including the bending margin.
- Bend the curved part.
- After the bending work is complete, remove any remaining bending margin.

Heating margin

Extra material applied to the manufacturing part to ensure proper object shape and size after heating.

Heating margin is a margin applied to a part that is bent by heating, or to a part that is known to distort during the manufacturing process and requires heating to remove the distortion. The margin is used by the heating process. Heating margins are applied as oblique or parallel margins to the edge of the part. The workflow in the shop is as follows:

- Cut the part including the heating margin.
- Heat the part.

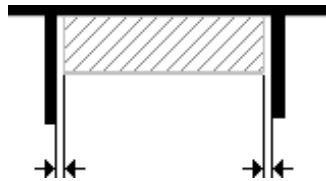
End face margin

Extra material applied to the manufacturing part that is consumed as part of the assembly process. Extra material is typically added parallel to the part end face or edge at the assembly or block lateral face.

During assembly, the margin gradually shrinks and eventually disappears after completion of the assembly or block.

Intercostal margin

Material that is *removed* at the ends of a manufacturing part when the ends are fillet welded.



Generic Margin - Non Trimmed

Extra material applied to the manufacturing part that is consumed during manufacturing.

Generic Margin - Trimmed

Extra material applied to the manufacturing part that should be removed after manufacturing.

Start Value

Specifies the amount of margin to apply at the identified start point of the selected lateral face. This option displays when **Oblique Margin** is selected in the **Mode** box.

End Value

Specifies the amount of margin to apply at the opposite point of the identified start point at the selected lateral face. The oblique margin is linearly distributed between the given start and end points. This option displays when **Oblique Margin** is selected in the **Mode** box.

Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.

Geometry Change

Indicates whether the margin should be trimmed. The following options are available:

- **As Margin** - Trimming is needed. The extension is only on the edge.
- **By Stretch** - No trimming is needed. The software stretches the shape along the part like shrinkage.
- **As Reference** - No request to extend the geometry. The margin is placed as a reference.

Stretch Offset

The length of the adjacent edge that the software adjusts while applying the margin. This is only editable if the **Geometry Change** is set to **By Stretch**.

Action

Indicates whether or not the margin properties are applied to the margin object. The possible values for objects that you are creating are **Apply** and **Ignore**. The possible values for objects that you are modifying are **Apply**, **Ignore**, and **Delete**. If the value is set to **Apply**, the modified properties are applied to the margin object. If the value is set to **Ignore**, then the modified properties are not applied to the margin object. If the value is set to **Delete**, then the margin object is deleted.

Rule

Indicates whether or not margins are placed by rule.

Type

Indicates the group margin type. This is the same value as shown in the ribbon bar.

Value

Indicates the group margin value. This is the same value as shown in the ribbon bar.


Update Dependent Objects


Indicates whether or not the dependent object parameters need to be updated if the parent margin properties change.

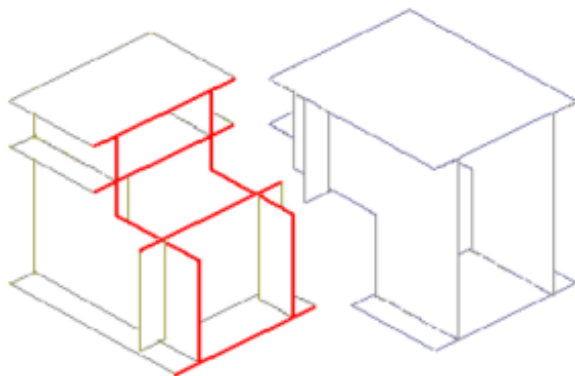
Add Margin

Adds a new row to the grid. This new row indicates the properties of another margin object that to create at the same port of the selected part. By default, all of the properties of the selected margin are copied onto the new margin object except for the margin type. You can edit these properties.

Margin By Assembly Connections

 Creates or modifies margins at the connections between two assemblies or blocks. It creates an assembly margin parent object grouping all the assembly margin children.

 **NOTE** For built-up profiles, you can define the margin for the web and flange separately.



Margin By Assembly Connections Ribbon

Displays the controls used to add a fabrication margin to all of the plate edges or profile ends connected to a selected assembly.



Assembly Margin Settings

Opens the **Assembly Margin Settings** dialog box, which you use to view and modify the properties of the fabrication margin that you are about to place in the model. For more information, see *Assembly Margin Properties Dialog Box* (on page 120).



Block or Assembly

Select the block or assembly to which you want to add the margin.



Adjacent Block or Assembly

Select the adjacent block or assembly to specify the connection between assemblies to which you want to add the margin.



Part

Select the plate or profile part to which you want to add the fabrication margin. You can select the part in a graphic view or in the **Workspace Explorer**.

NOTES

- Connected fabrication margins are always **Constant**.
- If the **Rule** box is not selected, then you can select any plates. The software applies the last used margin as a default.
- If you clear the selection from a part, then the software automatically clears the selection from the faces of that part.

**Margin Properties**

Displays a dialog box that allows you to review or edit the properties of the margins that you are creating or editing. For more information, see *Margin Properties Dialog Box (Margin By Assembly Connections)* (on page 117).

Finish

Creates the fabrication margin using the parameters that you have defined.

**Cancel**

Clears the selection.

**Accept**

Accepts the selection.

Name Rule

Specifies the naming rule for the margin objects that you are creating. The naming rule is not applied on margin objects that you are modifying.

Assembly Filter

Specifies the filter for selecting assemblies.

Rule

Specifies that the margin is placed by rule.

Type

Select the type of margin to apply. For more information on the available types, see *Fabrication Margin* (on page 93).

Value

Specifies the amount of margin to apply. This option displays when The **Mode** box is set to **Constant**.



Offset Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.


What do you want to do?

- *Add a margin by assembly connections* (on page 117)
 - *Modify an assembly margin* (on page 117)
 - *Delete an assembly margin* (on page 117)
-



Add a margin by assembly connections

1. Click **Margin By Assembly Connections** .
2. Select the assembly or block based on the filter criteria that you have defined using the **Assembly Filter** box.
3. Select the connected assembly or block based on the filter criteria that you have defined using the **Assembly Filter** box.
4. Click **Margin Properties** .
*The **Margin Properties** dialog box displays.*
5. Set the margin settings to meet your needs. For more information, see *Margin Properties Dialog Box (Margin By Assembly Connections)* (on page 117).
6. Click **Finish**.

Modify an assembly margin

1. Click **Select**  on the vertical toolbar.
 2. Select the assembly margin to modify.
TIP You can select the assembly margins from the **Workspace Explorer**.
 3. Using the ribbon controls, edit the margin as needed.
- NOTE** You can select multiple Assembly Margin or Assembly Margin Child objects to modify at the same time.

Delete an assembly margin

1. Click **Select**  on the vertical toolbar.
2. Select the assembly margin to delete.
3. Click **Delete** .

Margin Properties Dialog Box (Margin By Assembly Connections)

Name

Specifies the name of the part on which the margin is being created or modified. This column is not editable.

Status

Indicates whether the margin is being created or modified. This column is not editable.

Rule

Indicates whether the margin properties come from the rule or are user defined. The possible values are **Apply** and **Ignore**. By default, this value is set to **Apply** for all margins that are being created. If you change the values, the rule status automatically changes to **Ignore**. For

the existing margins, the status is **Ignore**. This column is editable.

Mode

Indicates the mode of the margin object. This column is not editable.

Type

Type of margin to apply. This column is editable.

Bending margin

Extra material applied to the manufacturing part to ensure proper object shape and size after bending.

Bending margin is a plate or profile part before bending. The margin is used during the bending work on the part using a bending machine. The bending margin is applied parallel to the edge of the part. You can apply bending margins to all sides of the part. The bending margin is removed after the bending work of the part is completed. The workflow in the shop is as follows:

- Cut the part including the bending margin.
- Bend the curved part.
- After the bending work is complete, remove any remaining bending margin.

Heating margin

Extra material applied to the manufacturing part to ensure proper object shape and size after heating.

Heating margin is a margin applied to a part that is bent by heating, or to a part that is known to distort during the manufacturing process and requires heating to remove the distortion. The margin is used by the heating process. Heating margins are applied as oblique or parallel margins to the edge of the part. The workflow in the shop is as follows:

- Cut the part including the heating margin.
- Heat the part.

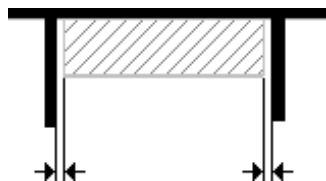
End face margin

Extra material applied to the manufacturing part that is consumed as part of the assembly process. Extra material is typically added parallel to the part end face or edge at the assembly or block lateral face.

During assembly, the margin gradually shrinks and eventually disappears after completion of the assembly or block.

Intercostal margin

Material that is *removed* at the ends of a manufacturing part when the ends are fillet welded.



Generic Margin - Non Trimmed

Extra material applied to the manufacturing part that is consumed during manufacturing.

Generic Margin - Trimmed

Extra material applied to the manufacturing part that should be removed after manufacturing.

Start Value

Specifies the amount of margin to apply at the identified start point of the selected lateral face. This option displays when **Oblique Margin** is selected in the **Mode** box.

End Value

Specifies the amount of margin to apply at the opposite point of the identified start point at the selected lateral face. The oblique margin is linearly distributed between the given start and end points. This option displays when **Oblique Margin** is selected in the **Mode** box.

Direction

Indicates the direction in which the margin is applied to the object. When the direction is set to **Perpendicular to Edge**, the margin is applied normal to the lateral face port. When the direction is set to **Global** for a plate, the margin is applied in the direction along the X, Y, or Z axis in which the plate is oriented. If the direction is set to **Global** for a profile, the margin is applied along the landing curve direction.

Geometry Change

Indicates whether the margin should be trimmed. The following options are available:

- **As Margin** - Trimming is needed. The extension is only on the edge.
- **By Stretch** - No trimming is needed. The software stretches the shape along the part like shrinkage.
- **As Reference** - No request to extend the geometry. The margin is placed as a reference.

Stretch Offset

The length of the adjacent edge that the software adjusts while applying the margin. This is only editable if the **Geometry Change** is set to **By Stretch**.

Action

Indicates whether or not the margin properties are applied to the margin object. The possible values for objects that you are creating are **Apply** and **Ignore**. The possible values for objects that you are modifying are **Apply**, **Ignore**, and **Delete**. If the value is set to **Apply**, then the modified properties are applied to the margin object. If the value is set to **Ignore**, then the modified properties are not applied to the margin object. If the value is set to **Delete**, then the margin object is deleted.

Rule

Indicates whether or not margins are placed by rule.

Type

Indicates the group margin type. This is the same value as shown in the ribbon bar.

Value

Indicates the group margin value. This is the same value as shown in the ribbon bar.

Offset Direction

Indicates the offset direction. This is the same value as shown in the ribbon bar.

Update Dependent Objects

Indicates whether or not the dependent object parameters need to be updated if the parent margin properties change.

Add Margin

Adds a new row to the grid. This new row indicates the properties of another margin object to create at the same port of the selected part. By default, all of the properties of the selected margin are copied onto the new margin object except for the margin type. You can edit these properties.

Assembly Margin Properties Dialog Box

Specifies the properties for the assembly margin that you are editing.

See Also

General Tab (Assembly Margin Properties Dialog Box) (on page 120)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Assembly Margin Properties Dialog Box)

Name

Specifies the name of the assembly margin.

Name Rule

Select the naming rule to use to name the assembly margin. Select **User Defined** to type a name in the **Name** box.

Plate Margin

Specifies the margin value to add to plates that are connected in the adjacent block or assembly.


Profile Margin

Specifies the margin value to add to profiles that are connected in the adjacent block or assembly.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its

relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Assembly Margin Child Properties Dialog Box

Specifies the properties for the assembly margin child that you are editing.

See Also

General Tab (Assembly Margin Child Properties Dialog Box) (on page 121)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Assembly Margin Child Properties Dialog Box)

Name

Specifies the name of the assembly margin child.

Name Rule

Select the naming rule to use to name the assembly margin child. Select **User Defined** to type a name in the **Name** box.

Plate Margin

Specifies the margin value to add to plates that are connected in the adjacent block or assembly.

Profile Margin


Specifies the margin value to add to profiles that are connected in the adjacent block or assembly.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using

Manufacturing Service Manager . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

NOTE For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Mode

Specifies the margin mode of the assembly margin child.

Type

Specifies the margin type of the assembly margin child.

Value

Specifies the applied margin value.

Direction

Specifies the direction in which the margin is applied.


Geometry Change

Specifies whether or not the margin should be trimmed.

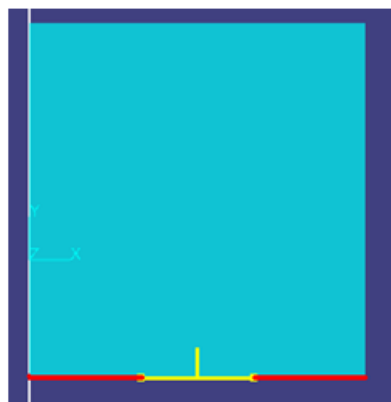
Stretch Offset

Specifies the stretch offset value set on the assembly margin child.

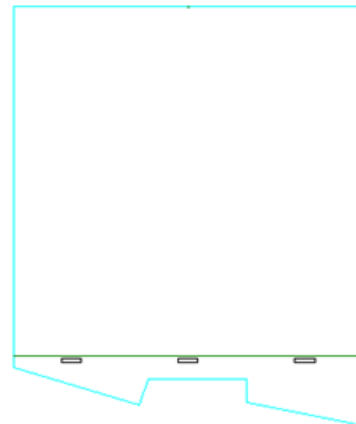
Regional Margin

 Adds a margin over a region of an edge. You can create multiple regional margins along a single edge.

The following example shows two constant margins (displayed in red) and an oblique margin (displayed in yellow) on the plate.



Model



Part Monitor Output

The following example shows a constant margin of -20mm on a profile.



NOTES

- You can provide an offset type and distance when using the **Along Edge** selection method. The software calculates the region based on these properties.
- You can select a reference plane or surface that does not intersect the selected port when using the **Reference** selection method. The software places the point on the port after calculating the offset.
- You can select plates, profiles, seams, and so on that intersect the selected port and provide offset properties when using the **Reference** selection method. The software calculates the margin end point based on the point of intersection and offset properties.

Regional Margin Ribbon

Controls parameters for adding a regional margin.

Properties

Activates the **Regional Margin Properties** dialog box, which you use to view and modify the settings for the regional margin you have selected. For more information, see *Regional Margin Properties Dialog Box* (on page 127).

Select Part

Select the object on which to place the margins. The object can be a plate, profile, or member, depending on the setting in the **Part Filter** box. This option is not available during modification.

Select Edge

Select the edge of the object on which to place the margins. This option is not available during modification.

Select Edge Segment

Select the edge segment upon which to place the regional margin. This option is only available for profiles and members.

Select Start Point

Select the start point of the margin. This can be either of the two end points of the edge segment.

 **Select End Point**

Select the end point of the margin.

Finish

Creates or modifies the regional margin.

 **Cancel**

Clears the selection.

 **Accept**

Accepts the selection.

Part Filter

Indicates which object can be selected. Available options include:

- **Plate**
- **Profile**
- **Member**





Rule

Indicates whether the margin is created by rule.











Sel Method

Indicates the method for selecting the end points of the region on which the margin is placed.

Available options for profiles or members include:

- **Edge Segment** - Select the edge segment of the profile or member edge. The ribbon displays the **Select Edge**  and **Select Start Point**  options.
- **Along Edge** - Select a point anywhere along the edge. The ribbon displays the **Select Start Point**  and **Select End Point**  options.

Available options for plates include:

- **Along Edge** - Select a point anywhere along the edge. The ribbon displays the **Select Start Point**  and **Select End Point**  options.
- **Reference Object** - Select a point at a distance from the selected part. The ribbon displays the **Select Start Point**  and **Select End Point**  options.
- **Seam/Knuckle** - Select a seam or knuckle intersecting the selected edge. The ribbon displays the **Select Start Point**  and **Select End Point**  options.
- **Start-Mid-End** - Select start, mid, or end points of the selected edge. The ribbon displays the **Select Start Point**  and **Select End Point**  options.
- **Features** - Select a feature along the selected edge. The ribbon displays the **Select Start Point**  and **Select End Point**  options.
- **Offset Type** - Specifies the direction in which the offset is applied. Available options include:
 - **Girth**
 - **Vertical**

- **Longitudinal**
- **Transversal**
- **Offset Value** - Specifies the amount by which to offset from the selected reference object.

Mode

Specifies the mode of margin to apply. You cannot change the mode of an existing margin. Available options include:

- **Constant**
- **Oblique**

Type

Specifies the type of margin to apply. Available options include:

- **Heating**
- **Bending**
- **Intercostal**
- **Coaming**
- **End Face**
- **Non-trimmed**
- **Trimmed**

Start Value

Specifies the amount of margin to apply. If **Mode** is set to **Constant**, this is the margin value. If **Mode** is set to **Oblique**, this is the margin value applied to the start end.

End Value

Specifies the amount of margin to apply to the end point when **Mode** is set to **Oblique**.



Add

Creates a new margin, or updates the selected margin. The new margin highlights in white.



Delete

Deletes the selected margin. This option is only available when you select one of the margins in the **Show Form** dialog box.

Form



Indicates whether to display a dialog box containing all of the margins that have been placed. All of the margins in this dialog box are created when you click **Finish**. Select a row to display the corresponding margin values in the ribbon bar. When you select a margin on the dialog box, the corresponding margin highlights in the model.

Constant	Heating	.10 m	.10 m
Oblique	Heating	.20 m	.30 m
Constant	Heating	.30 m	.30 m




What do you want to do?

- *Add a regional margin* (on page [126](#))
 - *Modify a regional margin* (on page 126)
 - *Delete a regional margin* (on page 127)
-



Add a regional margin

1. Click **Regional Margin** .
*The **Regional Margin** ribbon displays.*
2. Select the type of object to which to apply the margin from the **Part Filter** list.
3. Select the part to which to apply the margin from the model.
4. Select the edge to which to apply the margin.
5. Select the selection method from the **Sel Method** list.
6. Depending on the selection method you chose, select the start and end points or the edge segment and start point.
7. Set up the remaining properties to meet your requirements.
8. Click **Properties** .
*The **Regional Margin Properties** dialog box displays.*
9. Set the margin settings to meet your needs. For more information, see *Regional Margin Properties Dialog Box* (on page 127).
10. Click **Finish**.

Modify a regional margin

1. Click **Select**  on the vertical toolbar.
2. Select the regional margin to modify.
 **TIP** You can select the margins from the **Workspace Explorer**.
*The margins highlight in the model. All margins on the same edge as the selected margin display in the **Show Form** dialog box.*
3. Using the ribbon controls, edit the margin as needed. For more information, see *Regional Margin* (on page 122).
4. Click **Add**  to update the changes to a margin.
5. Click **Finish**.

Delete a regional margin

1. Click **Select**  on the vertical toolbar.
2. Select the margin to delete.
3. Click **Delete** .

Regional Margin Properties Dialog Box

Specifies the properties for the regional margin that you are editing.

See Also

General Tab (Regional Margin Properties Dialog Box) (on page 127)

Feature Tab (Regional Margin Properties Dialog Box) (on page 128)

General Tab (Regional Margin Properties Dialog Box)

Name

Specifies the name of the regional margin.

Name Rule

Select the naming rule to use to name the regional margin. Select **User Defined** to type a name in the **Name** box.

Direction

Specifies the common margin direction for the margin objects.

Geometry Change

Specifies the common margin geometry change for the margin objects.

Stretch Offset

Specifies the common margin stretch offset value for the margin objects.

Mode

Specifies the mode of margin to apply. You cannot change the mode of an existing margin. Available options include:

- **Constant**
- **Oblique**

Type

Specifies the type of margin to apply. Available options include:

- **Heating**
- **Bending**
- **Intercostal**
- **Coaming**
- **End Face**

- **Non-trimmed**
- **Trimmed**

Start Value

Specifies the amount of margin to apply. If **Mode** is set to **Constant**, this is the margin value. If **Mode** is set to **Oblique**, this is the margin value applied to the start end.

End Value

Specifies the amount of margin to apply to the end point when **Mode** is set to **Oblique**.

Feature Tab (Regional Margin Properties Dialog Box)

Displays all of the features on the edge of the plate. If there are no features, this tab is blank. You can assign the **Fix** or **Move** property for each margin. This property is then common to all of the margins you place.

SECTION 4


Shrinkage

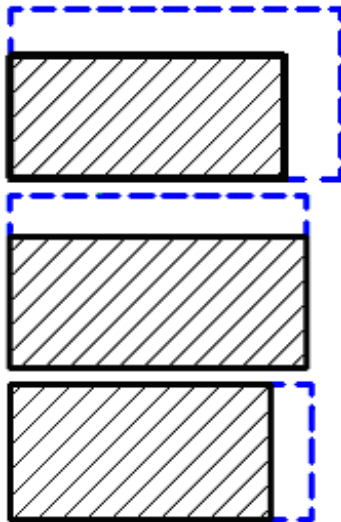
The shrinkage commands stretch manufacturing parts to account for steel shrinkage during construction.

In This Section

Shrinkage	129
Multi Shrinkage.....	137

Shrinkage

 Stretches a manufacturing part to account for steel shrinkage during welding. Adding shrinkage changes the position of marks and openings on the part because they are stretched along with the part. The part shrinks back to its needed size and shape in the welding process.




For plate parts, you can define shrinkage in one or two directions as scaling shrinkage (defined by the expansion ratio and the expansion direction). For profiles, you can only apply scaling shrinkage to the length.

By default, the software prompts you to identify the edges so that it can determine the shrinkage directions. If you select the shrinkage type by global axis, the software sets the primary and secondary axis based on the plate type:

- For deck plates, the primary factor is the X axis and the secondary factor is Y axis.
- For transverse plates, the primary factor is the Y axis and the secondary factor is the Z axis.
- For longitudinal plates, the primary factor is the X axis and the secondary factor is the Z axis.

A small coordinate system, based on the shrinkage directions with the appropriate values, also displays in the Part Monitor.

 **NOTE** For built-up profiles, you can define the shrinkage for the web and flange separately.

Shrinkage Ribbon


Displays the controls used to add a shrinkage margin to plate edges or profile ends.

Properties

Activates the **Shrinkage Properties** dialog box, which you use to view and modify the properties of the shrinkage margin that you are about to place in the model. For more information, see *Shrinkage Properties Dialog Box* (on page 135).

Plates or Profiles

Select the plate or profile part to which you want to add the shrinkage. You can select the part in a graphic view or in the **Workspace Explorer**.

 **NOTE** Because of the different ways in which shrinkage is applied to plates and profiles, you can only select either plates or profiles.

Primary Direction

Select a line to indicate the primary direction for the shrinkage. The primary direction vector is in the direction of the first selected plate lateral face.

Secondary Direction

Select a line to indicate the secondary direction for the shrinkage. The secondary direction vector is perpendicular to the primary direction vector unless another plate lateral face is selected.

Finish

Creates the shrinkage using the parameters that you have defined.

Rule

Check the rule to get values of **Shrinkage Type**, directions (**Primary** and **Secondary**), and factors (**Primary**, **Secondary**, and **Tertiary**) from the rule for the selected part to place shrinkage.

Mode

Select the mode of shrinkage to apply. The following options are available:

- **Assembly** - Places global axis shrinkage on an assembly.
- **Part-Assembly Based** - Places shrinkage on the parts of an assembly depending on the shrinkage of the assembly.
- **Part-Private** - Places shrinkage on the parts.

Shrinkage Type

Select the direction type of shrinkage to apply.

- **By Axis** - Uses a grid system axis to define the shrinkage direction.
- **By Edge** - Uses one of two edges to determine the shrinkage directions.
- **By Vector** - Provides an additional list called selection method from which you can select the primary and secondary shrinkage directions.

Tertiary Factor

Select a factor for the amount of extra material to apply in the tertiary direction.

Frame System

Shows all the frame systems from the catalog rule in the range of the selected plate. This option is only visible when the **Type** is **By Axis**.

Primary Direction

Select an axis corresponding to the frame system selected in the **Frame System** select list to represent the Primary Direction. This axis can also be picked in the **Primary Direction** step. This option is only visible when the **Type** is **By Axis**.

Secondary Direction


Select an axis corresponding to the frame system selected in the **Frame System** select list to represent the Secondary Direction. This axis can also be picked in the Secondary Direction step. This option is only visible when the **Type** is **By Axis**.


Primary Factor

Select a factor for the amount of extra material to apply in the primary direction.

Secondary Factor

Select a factor for the amount of extra material to apply in the secondary direction.



 **NOTE** The units displayed in the primary and secondary factor fields are determined by the **Slope** setting on the **Units of Measure** tab of the **Options** dialog box. Click **Tools > Options** to display this dialog box.

 **IMPORTANT** If no shrinkage is needed in the secondary direction, use a **Secondary Factor** of 1.00.

What do you want to do?



- *Add scaling by edge shrinkage* (on page [132](#))
 - *Add scaling by axis shrinkage* (on page 132)
 - *Modify shrinkage* (on page 134)
 - *Copy and paste shrinkage* (on page 134)
 - *Delete shrinkage* (on page 135)
-


Add scaling by edge shrinkage

1. Click **Shrinkage** .
2. On the **Shrinkage** ribbon, select the **Mode** that meets your requirements.
3. Click **Plates/Profiles/Assembly** .
4. Select the parts to which you want to add shrinkage.

TIPS



- You can select the parts in the model, or from the **Workspace Explorer**.
- Because of the different ways in which shrinkage is applied to plates and profiles, you can only select one type at a time.

5. Click **Primary Direction** .
6. Select a line to indicate the primary direction for the shrinkage.
7. Click **Secondary Direction** .
8. Select a line to indicate the secondary direction for the shrinkage.
9. Type a value in the **Primary Factor** box.
10. Type a value in the **Secondary Factor** box.

 **IMPORTANT** If no shrinkage is needed in the secondary direction, then a Secondary Factor of 1.00 should be used.

11. Click **Finish**.

Add scaling by axis shrinkage

1. Click **Shrinkage** .
2. On the **Shrinkage** ribbon, select the **Mode** that meets your requirements.
3. Click **Plates/Profiles/Assembly** .
4. Select the parts to which to add shrinkage.

TIPS

- You can select the parts in the model or from the **Workspace Explorer**.
- Because of the different ways in which shrinkage is applied to plates and profiles, you can only select one type at a time.

5. Select **By Axis** from the **Shrinkage Type** list.
6. Select a grid system to define the shrinkage directions.
7. Select the primary axis from the list.
8. Select the secondary axis from the list.
9. Type a value in the **Primary Factor** box.
10. Type a value in the **Secondary Factor** box.



★ **IMPORTANT** If no shrinkage is needed in the secondary direction, use a **Secondary Factor** of 1.00.

11. Type a value in the **Tertiary Factor** box.

★ **IMPORTANT** If no shrinkage is needed in the tertiary direction, then use a **Tertiary Factor** of 1.00.

12. Click **Finish**.

Add by vector shrinkage

1. Click **Shrinkage** .
2. On the **Shrinkage** ribbon, select the **Mode** that meets your requirements.
3. Click **Plates/Assembly** .
4. Select the parts to which to add shrinkage.

💡 TIPS

- You can select the parts in the model or from the **Workspace Explorer**.
- Because of the different ways in which shrinkage is applied to plates and profiles, you can only select one type at a time.

5. Select **By Vector** from the **Shrinkage Type** list.
6. Select the selection method from the **Selection Method** list.
7. Depending on the selection method you chose, select the Primary and Secondary directions.
8. Type a value in the **Primary Factor** box.
9. Type a value in the **Secondary Factor** box.


★ **IMPORTANT** If no shrinkage is needed in the secondary direction, use a **Secondary Factor** of 1.00.

10. Type a value in the **Tertiary Factor** box.

★ **IMPORTANT** If no shrinkage is needed in the tertiary direction, then use a **Tertiary Factor** of 1.00.

11. Click **Finish**.


Modify shrinkage

1. Click **Select**  on the vertical toolbar.
2. Select the shrinkage to modify.
TIP You can select the shrinkage in the model, or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the shrinkage as needed.

NOTES

- You cannot change the parent plate when you modify the object.
- If you are modifying a single plate, you can change the shrinkage type.
- If you set the value of the **Shrinkage Type** box to **By Axis**, then the software computes both the primary and secondary axes automatically.



Copy and paste shrinkage

1. Click **Select**  on the vertical toolbar.
2. Select the shrinkage to copy.
TIPS
 - You can select the shrinkage in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Shrinkage** to make the selection easier.
3. Click **Edit > Copy**.
4. Click **Edit > Paste**.
5. Select the part on which to paste the shrinkage.

TIPS

- You must select a destination that does not already have shrinkage.
 - If the shrinkage that you copied is from a plate, you must select a plate for the destination. If the shrinkage that you copied is from a profile, you must select a profile for the destination.
6. Select the primary direction for the shrinkage.
 7. Optionally, select a secondary direction for the shrinkage.
 8. Click **OK** on the **Paste** dialog box.

Delete shrinkage

1. Click **Select**  on the vertical toolbar.
2. Select the shrinkage to delete.
3. Click **Delete** .

Shrinkage Properties Dialog Box

Specifies the properties for the shrinkage that you are editing.

See Also

General Tab (Shrinkage Properties Dialog Box) (on page 135)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

Shrinkages Tab (Shrinkage Properties Dialog Box) (on page 137)

General Tab (Shrinkage Properties Dialog Box)

Name

Specifies the name of the shrinkage.

Name Rule

Select the naming rule to use to name the shrinkage.

- Select **DefaultNameRule** to let the software automatically name new objects. If the Shrinkage Type is set to Scaling by Edge, the following syntax is used: <"Scaling shrinkage ">-<Workshare Location ID>. For example, Scaling shrinkage -1. If the Shrinkage Type is set to Global Axis, the following syntax is used: <"Global shrinkage ">-<Workshare Location ID>. For example, "Global shrinkage -1.
- Select **User Defined** to type a name in the **Name** box.

Detailed Part Status

Displays the status of the parent object for the shrinkage. This property only displays when you are modifying a shrinkage.

Manufacturing Object Status

Displays the status of the shrinkage. This property only displays when you are modifying a shrinkage.

Shrinkage Type

Select the direction type of shrinkage to apply.

- Select **By Axis** to select a grid system axis to define the shrinkage direction.
- Select **By Edge** to select one of two edges so that it can determine the shrinkage directions.
- Select **By Vector** and the software provides one more list called selection method and depends upon the selection method there you can select the primary and secondary directions.

Primary Direction

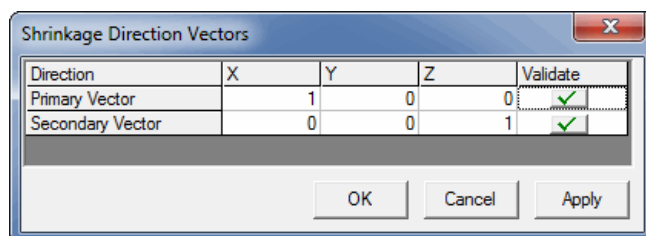
Displays the coordinate system name and primary direction of the shrinkage when the **Shrinkage Type** is **By Axis**. When **Shrinkage Type** is **By Vector** depends upon the selection method it will show axis (Coordinate System) or vector (Global and vector). This box is empty when **Shrinkage Type** is **By Edge** or selection method is **Edge**.

Secondary Direction

Displays the coordinate system name and secondary direction of the shrinkage when the **Shrinkage Type** is **By Axis**. When **Shrinkage Type** is **By Vector** depends upon the selection method it will show axis (Coordinate System) or vector (Global and vector). This box is empty when **Shrinkage Type** is **By Edge** or selection method is **Edge**.

Selection Method

- **Coordinate System** - Select primary and secondary direction vectors as axis.
- **Edge** - Select primary and secondary direction vectors as plate edges.
- **Global** - Primary and secondary direction as standard vectors [X (1, 0, 0), Y (0, 1, 0), Z (0, 0, 1)].
- **Vector** - User can create shrinkage in any direction with the help of Shrinkage Direction Vectors control.



- **Validate** calculates the primary/secondary vector with respect to the plate normal and primary/secondary vector.

Primary Factor

Select a factor for the amount of extra material to apply in the primary direction.

Secondary Factor

Select a factor for the amount of extra material to apply in the secondary direction.

Tertiary Factor

Select a factor for the amount of extra material to apply in the tertiary direction. Use values from 0mm to 1m for plates and profiles.

See Also

Shrinkage Properties Dialog Box (on page 135)

Shrinkages Tab (Shrinkage Properties Dialog Box)

Displays the shrinkages on all the parent assemblies so that you can review how the software calculates shrinkage values for part-assembly based shrinkage. This tab also displays the values of the selected shrinkage.

This tab is available if the **Shrinkage Mode** is **Part - Assembly Based**, or **Part - Private**.

Name

Displays the shrinkage name.

Mode

Displays the shrinkage mode. **Assembly** indicates that the values are associated with the parent assembly shrinkages.

Primary Direction

Displays the coordinate system name and primary direction of the shrinkage when the **Shrinkage Type** is **By Axis**. When **Shrinkage Type** is **By Vector** depends upon the selection method it will show axis (Coordinate System) or vector (Global and vector). This box is empty when **Shrinkage Type** is **By Edge** or selection method is **Edge**.

Secondary Direction

Displays the coordinate system name and secondary direction of the shrinkage when the **Shrinkage Type** is **By Axis**. When **Shrinkage Type** is **By Vector** depends upon the selection method it will show axis (Coordinate System) or vector (Global and vector). This box is empty when **Shrinkage Type** is **By Edge** or selection method is **Edge**.

Primary Factor (mm/m)

Displays the primary factor of the shrinkage.

Secondary Factor (mm/m)

Displays the secondary factor of the shrinkage.

Tertiary Factor (mm/m)

Displays the tertiary factor of the shrinkage.

Multi Shrinkage



Stretches multiple manufacturing parts to account for steel shrinkage during welding.

Multi Shrinkage Ribbon

Specifies parameters for applying shrinkage to multiple parts.

Select

Select the objects that meet your criteria. You can select objects in the graphics view, or from the **Workspace Explorer**. Use the **Filter** list to limit the objects that are available for your selection.

**Multiple Shrinkage Properties**

Displays the **Multiple Shrinkage Properties** dialog box. For more information, see *Multiple Shrinkage Properties Dialog Box* (on page 140).

Finish

Creates or modifies the shrinkage using the parameters that you have defined.

**Accept**

Accepts the selected objects.

**Cancel**

Cancels the object selection.

Mode

Specifies the mode for the shrinkage objects that are created. The software changes the contents of the **Filter** list based on this option.

Filter

Specifies the filter criteria for selection. The available options are:

- **Parts** - Allows you to pick plate parts, profile parts, or member parts.
- **Parts or Assemblies** - Allows you to pick plate parts, profile parts, member part, or assemblies. If you select an assembly, the software automatically selects all parts of that assembly.
- **Plate Parts** - Allows you to pick plate parts.
- **Plate Part and Connected Profiles** - Allows you to pick plate parts. The software automatically selects any profiles connected to the plate parts.
- **Profile Parts** - Allows you to pick profile parts.
- **Member Parts** - Allows you to pick member parts.

Factor

Specifies the rule that the software uses while creating the shrinkage objects. You can modify this rule if necessary.


Name Rule

Specifies the naming rule for the shrinkage objects that are *created*. The naming rule is not applied to the shrinkage objects that are *modified* using this command.


What do you want to do?

- *Create shrinkage on multiple objects* (on page 139)
 - *Modify shrinkage on multiple objects* (on page 139)
 - *Delete shrinkage from multiple objects* (on page 139)
-


Create shrinkage on multiple objects

1. Select the shrinkage mode from the **Mode** list.
2. Select the value from the **Filter** list that best meets your requirement.
3. Select the objects to which you want to add shrinkage.
4. Select the shrinkage rule from the **Factor** list that proposes the values for shrinkages to place on the selected objects.
5. Click **Multiple Shrinkage Properties** .
6. Review and reset the parameters as required.
7. Set the **Action** to **Apply**.
8. Click **OK** on the **Multiple Shrinkage Properties** dialog box.
9. Click **Finish**.

Modify shrinkage on multiple objects

1. Select the shrinkage mode from the **Mode** list.
2. Select the value from the **Filter** list that best meets your requirement.
3. Select the objects for which you want to modify shrinkage. You can also select an assembly to add all its children to the **Multiple Shrinkage Properties** dialog box.
4. Click **Multiple Shrinkage Properties** .
5. Review and reset the parameters as required.
6. Set the **Action** to **Apply**.
7. Click **OK** on the **Multiple Shrinkage Properties** dialog box.
8. Click **Finish**.

Delete shrinkage from multiple objects

1. Select the value from the **Filter** list that best meets your requirement.
2. Select the objects from which you want to delete shrinkage.
3. Click **Multiple Shrinkage Properties**  and set the parameters to meet your requirements.
4. Set the **Action** to **Delete**.
5. Click **OK** on the **Multiple Shrinkage Properties** dialog box.
6. Click **Finish**.

Multiple Shrinkage Properties Dialog Box

Specifies the parameters for shrinkage objects that are created or modified.

Name

Specifies the name of the part on which shrinkage is either being created or modified. This column is not editable.

Type

Specifies the type of the object. The possible values for this column are **Plate Part**, **Profile Part**, **Member Part**, **Assembly**, and **Block**. This column is not editable.

Status

Indicates if a shrinkage object is being created or modified. The possible values for this column are **Create** or **Modify**. This column is not editable.

Mode

Indicates the mode of the shrinkage object. The possible values are **Assembly**, **Part-Assembly Based**, and **Part-Private**.

Factor Rule

Indicates the shrinkage rule.

Shrinkage Type

Indicates the shrinkage type of the shrinkage object. Possible values are **By Axis**, **By Edge**, and **By Vector**. For plate parts, only the **By Axis** and **By Vector** type of shrinkage can be placed using this command. If a rule proposes **Scaling by Edge** for a particular plate part, then the entire row would be disabled and shrinkage is not created for that plate part. However, you can modify the **Primary** and **Secondary** values of the scaling by edge shrinkage on a plate part. This column is not editable. For assemblies, you can place only the **By Axis** and **By Vector** type of shrinkage.

Frame System

Specifies the frame system from which the primary and the secondary directions of the shrinkage are selected. This column is editable. Possible values are all of the frame systems that are in range of the plate part and those given by the catalog rule.

Primary Direction

Specifies the primary direction of the shrinkage object. This column is editable.

Primary Value

Specifies the primary value of the shrinkage object. This column is editable.

Secondary Direction

Specifies the secondary direction of the shrinkage object. This column is editable.

Secondary Value

Specifies the secondary value of the shrinkage object. This column is editable.

Tertiary Value

Specifies the tertiary value of the shrinkage object on an assembly. This column is editable for **Shrinkage with Assembly Mode**.

Action

Indicates whether the shrinkage properties are applied to the shrinkage object or not. The possible values in creation are **Apply** and **Ignore**. Possible values in modification are **Apply**, **Ignore**, and **Delete**. If this option is set to **Apply**, then the modified properties are applied to the shrinkage object. If it is set to **Ignore**, then the modified properties are not applied to the shrinkage object. If this option is set to **Delete**, the shrinkage object is deleted.

Filters

Defines a filter for the displayed rows. You can use these options to view all of the existing shrinkages under assemblies, regardless of their type. The options from left to right are:

- **Mode** - Displays in italics any shrinkages that are of a mode different than the mode selected in the ribbon bar.
- **Object** - Refines the filter based on the type of objects that you select.
 - **Show All objects** - Displays all objects.
 - **Show Rule Objects** - Refines the filter based on the rule displayed in the **Rule** box.
 - **Show Out of Date Objects** - Displays objects that are out of date.
- **Rule** - Refines the filter based on the selected rule. This list contains all of the associated rules. This option is not available if the **Object** filter is set to **Show All Objects**.

Recompute

The list at the bottom of the dialog box contains options that determine which shrinkage objects display on the dialog box.

- **Show All Objects** - Displays all the objects from the selected list.
- **Show Rule Objects** - Displays all the objects for which the selected factor rule proposes eligible values.
- **Show Out of Date** - Displays all of the objects that are out of date.

Click **Recompute** to recalculate the values for each shrinkage object displayed on the dialog box. To apply the new values on the shrinkages, click **OK**, and **Finish** the command.

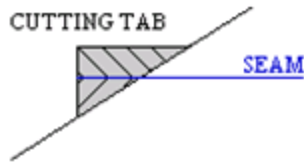
SECTION 5

Manufacturing Tab

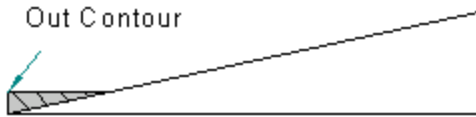


Adds manufacturing tabs to detailed parts at plate edges. You can place tabs of various types.

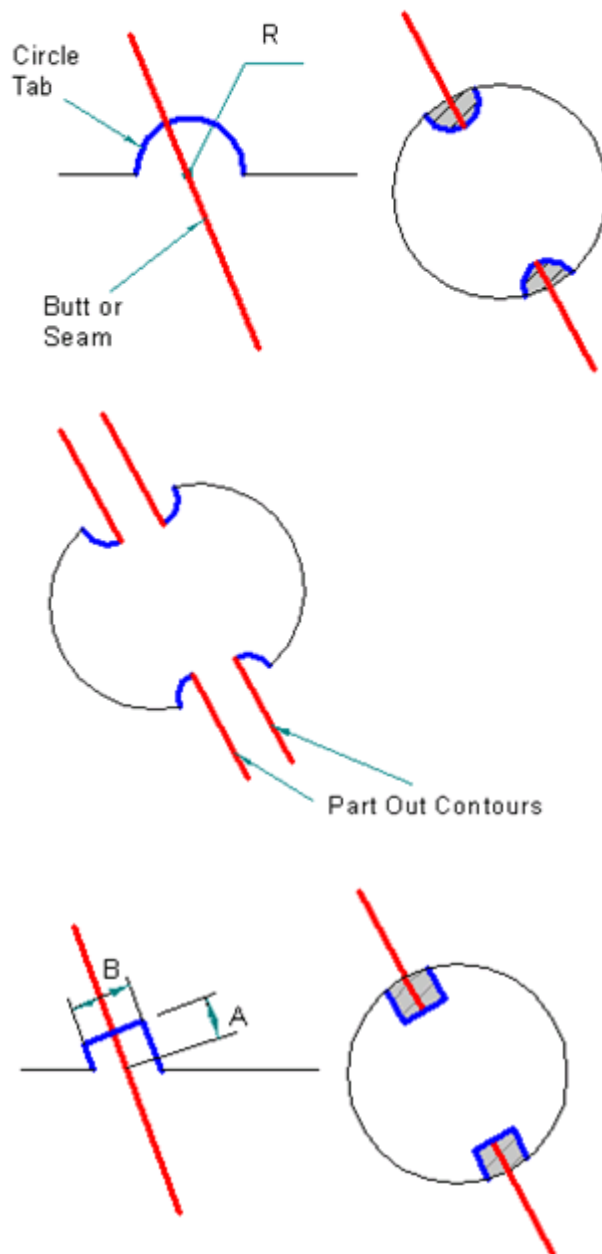
Plates may have sharp corners or bent edges that could become deformed during fabrication, erection or assembly. Tabs are additional material around such areas. Tabs are applied when seams or plate edges cross at angles that would produce acute angle corners. Welding often damages or melts off these sharp corners, so the tabs are added to protect the sharp part corners. These tabs are trimmed off after welding.



Tabs are introduced when cutting out a plate from a flat sheet, or padding material that is added later. Either way, tabs modify the manufactured part's outer contour.



A tab could have several shapes, depending upon where they are located. For example, in order to protect a circular opening through which a seam passes, you could protect the opening edge with tabs of different shapes as shown below.



Manufacturing Tab Ribbon

Displays the controls used to create or edit manufacturing tabs.



Properties

Activates the **Manufacturing Tab Properties** dialog box, which you use to view and modify the properties of the manufacturing tabs that you are about to place in the model. For more information, see *Manufacturing Tab Properties Dialog Box* (on page 146).

 **Plate Part**

Select the plate part on which to create the manufacturing tab.

 **Driven Part**

Select the port whose geometry changes as the tab is created.

 **Driving Object**

Select the object that is the entity that needs protecting. The geometry of the driving object may be lengthened a bit in the process of creating the manufacturing tab, but it is likely to remain largely unchanged.

 **Tab Position**

Select the position for the tab. This button is enabled when the software identifies an ambiguity while creating the tab.

Finish

Creates the marking line that you have defined.



 **Cancel**

Clears the selection.

 **Accept**

Accepts the selection.

Select Part Item

Specifies the filter for items that you can select when using the  **Driven Part** or  **Driving Object** options. Select **More** to display the **Select Tab Type** dialog box. You can select an item type from the catalog on this dialog box. For more information, see *Select Tab Type Dialog Box* (on page 146).

Retain Inputs



Specifies which inputs to the manufacturing tab Smart 3D retains.

- NoObjects
- Plate
- Plate, Driven
- Plate, Driven, Driving



What do you want to do?

- *Create a manufacturing tab* (on page 145)
 - *Modify a manufacturing tab* (on page 145)
 - *Delete a manufacturing tab* (on page 145)
-



Create a manufacturing tab

1. Click **Manufacturing Tab** .
2. Select the plate part.
3. Select a part item from the **Select Part Item** list. If the part that you want to use is not in the list, select **More** and then select a part from the **Select Tab Type** dialog box.
4. Select a driven object. The driven object is the object that changes to accommodate the tab. The objects that you can select are controlled by the value in the **Select Part Item** box.
5. Select a driving object. The driving object is the object that needs protection. The objects that you can select are controlled by the value in the **Select Part Item** box.
6. If the  **Tab Position** button is enabled, click it and select a point for the tab. This button is enabled if the driving object and the driven object intersect in more than one place.
7. Click **Finish**.

Modify a manufacturing tab

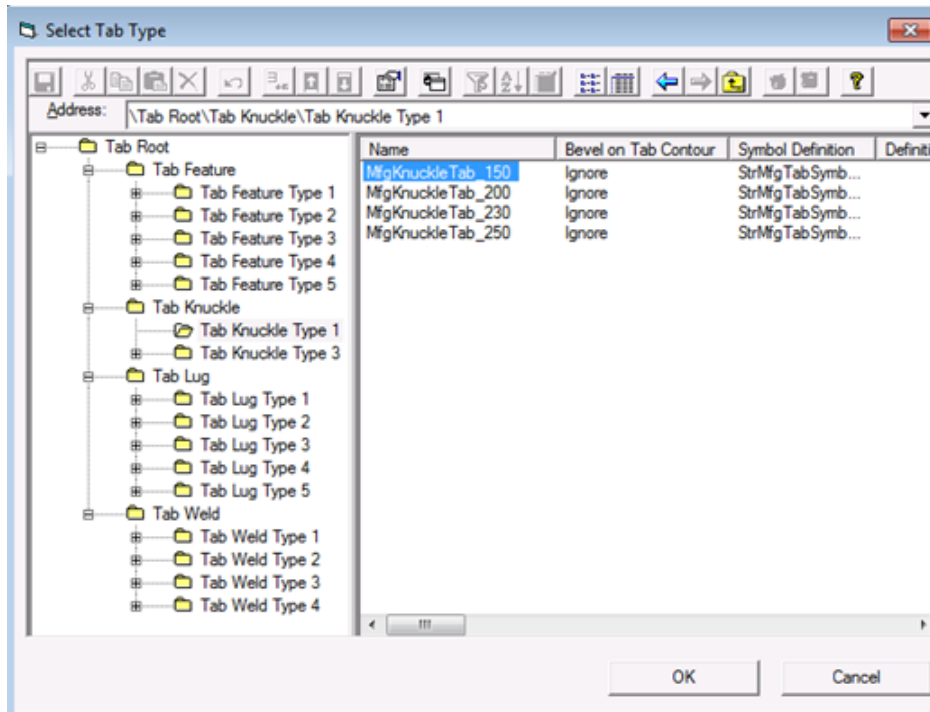
1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing tab to modify.
 **TIP** You can select the manufacturing tab from the **Workspace Explorer**.
3. Using the ribbon controls, edit the manufacturing tab as needed.

Delete a manufacturing tab

1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing tab to delete.
3. Click **Delete** .

Select Tab Type Dialog Box

Displays options for selecting manufacturing tab objects from the catalog.



Manufacturing Tab Properties Dialog Box

Specifies the properties for the manufacturing tab that you are editing.

See Also

General Tab (Manufacturing Tab Properties Dialog Box) (on page 146)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Manufacturing Tab Properties Dialog Box)

Name

Specifies the name of the manufacturing tab.

Name Rule

Select the naming rule to use to name the manufacturing tab.

- Select **DefaultNameRule** to let the software automatically name new objects. For a manufacturing tab, the following syntax is used: <"Mfg Tab">-<Workshare Location ID>. For example, "Mfg Tab-1".
- Select **User Defined** to type a name in the **Name** box.

Part Item

Indicates the name of the catalog part used to create the tab. This property is read only.

Detailed Part Status

Indicates the status of the detailed part. This property is read only.

Manufacturing Object Status

Indicates the status of the manufacturing object. This property is read only.

Bevel on Tab Contour

Indicates the bevel status on the outer contour created by the **Manufacturing Tab** command.

- **Ignore** - The bevel is not applied.
- **Driven** - The bevel is applied to the portion of the contour touching the driven edge.
- **Driving** - The bevel is applied to the portion of the contour touching the driving edge.
- **Both** - The bevel is applied to the portion of the contour touching either the driven or driving edges.

Bevel on Tab Mark

Indicates the bevel status on the marking between the plate and the tab.

- **Ignore** - The bevel is not applied.
- **Apply** - The bevel is applied along the marking.

SECTION 6

Templates

The template commands create a set of templates at user-defined positions.


NOTE When you create or edit a template, the dialog box title displays the plate part name using the format **Plate Part:** <name of the reference plane>.

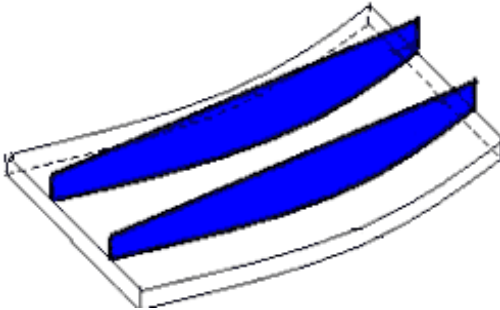
untitled.ses : Structural Detailing : Plant : Filter - [Plate Part : <NoBlk>-OLS.2-1]

In This Section

Plate Template	148
Profile Face Template	162
Tubular Member Template	168
Edge Template	177

Plate Template

 Creates a set of templates at user-defined positions. The command generates the template outer contour shapes and reference point data that can be used for output in template table data.



Templates are used to check and control the actual curvature during and after the bending.

A template is a single object aligned with a respective marking line on a plate. A template set is a group of templates prepared for a single plate to check its curvature during and after the bending process.

In addition to specifying the shape and spacing for the templates, you can apply rule-based processes and marking sets to the templates, analogous to the processes and markings applied to plate parts.

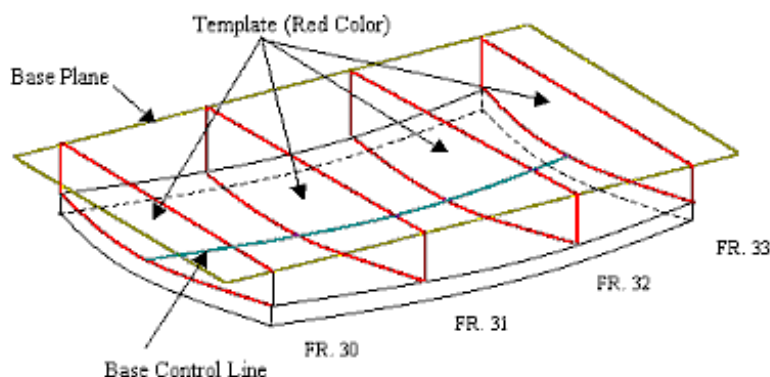
NOTE If the software cannot compute the base plane or the base control line by rule, it displays a message box to warn you of the condition.

The software identifies the following template types:

Frame

A frame template set is a general way to make a template. This type of template is used in most cases. A frame template set consists of templates at each frame position on a selected

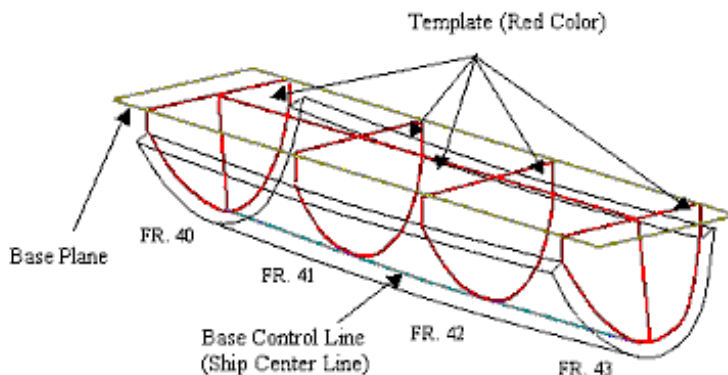
plate. You can specify an interval value for defining the template positions, and you can add additional templates if needed.



Aft and Forward

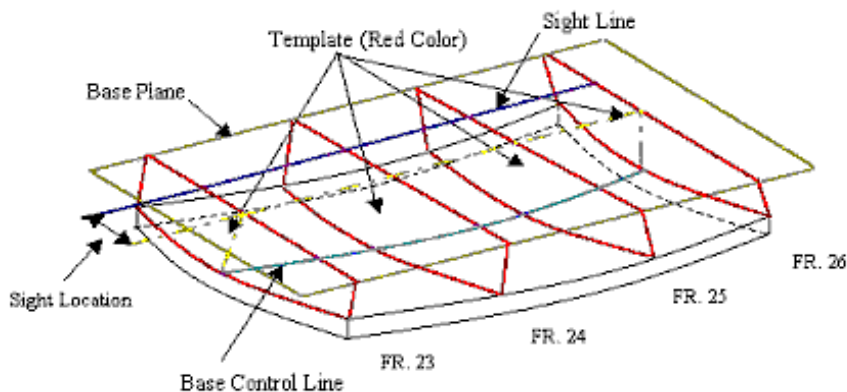
An aft and forward template set is similar to the frame template set, except that the location of the templates is determined by the aft and forward seam. **Center Line**

A center line template set is a special case of the frame template set where the base control line is a ship center line.



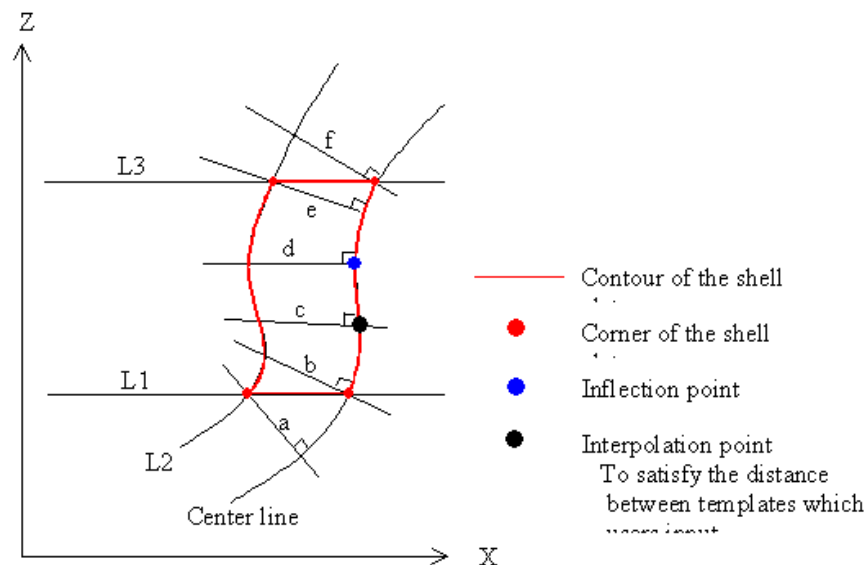
Perpendicular

A perpendicular template set specifies the templates at planes normal to the plate surface.

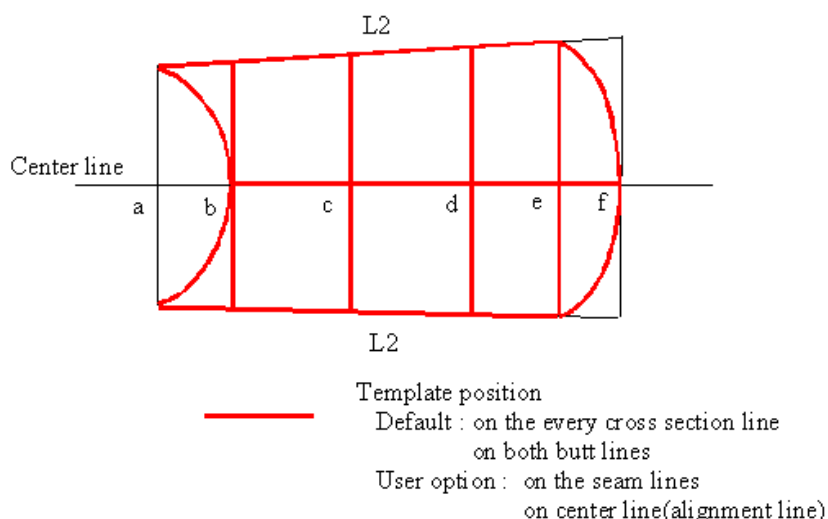


Stem/Stern

A stem/stern template specifies that the templates are set on the corner of the shell plate, the inflection points of the center line, and at interpolation points which are determined by the software to meet your input parameters. The software calculates intersections between the shell plate and the sections that are perpendicular to the centerline at each template position. The position of the inflection point, on the edge of a plate part, is determined by having the second derivate value of zero.



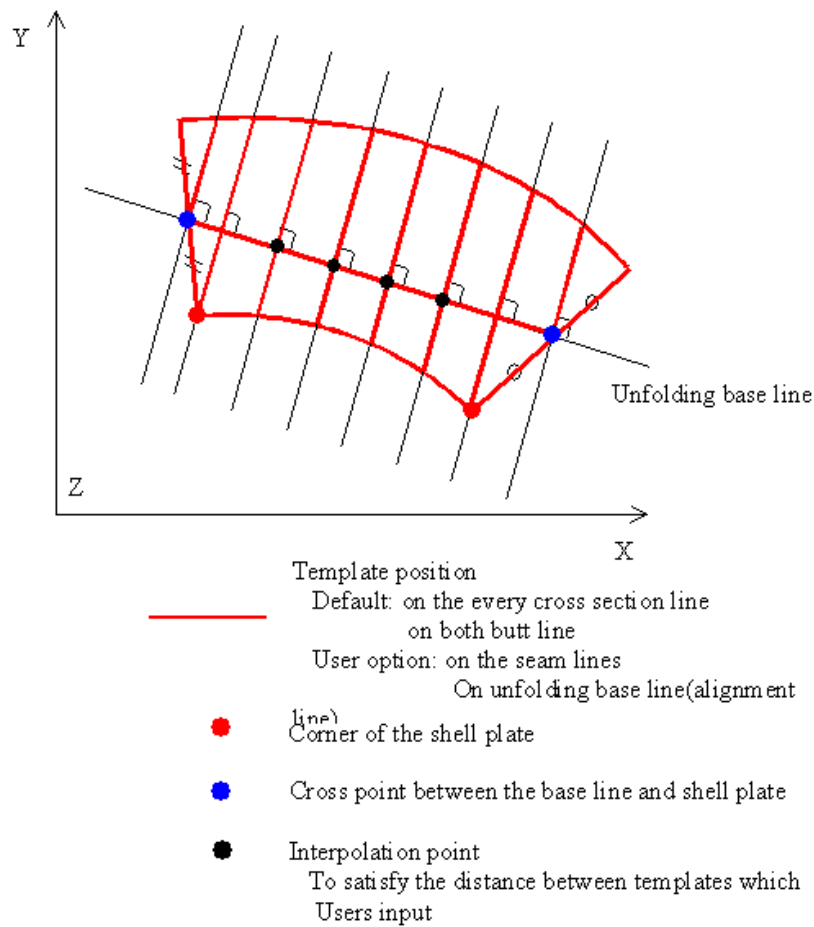
Unfolding



Perpendicular XY

A perpendicular XY template set is often positioned on the plates next to the stem/stern. The templates are set on the planes that are perpendicular to the XY plane and the unfolding base

line. The unfolding is defined by the midpoints of both butt lines. The software sets templates on the corner of the shell plate and interpolation points, which are determined by the software to satisfy your input parameters.



User Defined

A user defined template set lets you sketch the base control line and the template locations in the two-dimensional drafting environment. The template locations must intersect the base control line.

Plate Template Ribbon

Displays the controls used to add templates to the model.

Properties

Activates the **Template Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the template that you are about to place in the model. For more information, see *Plate Template Properties Dialog Box* (on page 156).

Plate Parts

Select the detailed plate part to which you want to add the template. You can select the part in

a graphic view or in the **Workspace Explorer**.

Rule

Indicates if you are using a customized filter to select the part.

Template Type

Indicates the template type on which to place the template. The available options are:

- **Plate** - Places templates on plate parts.
- **Profile Face** - Places templates on profile faces.



Base Plane

Selects the base plane for the template. The base plane is an offset balance plane of a plate or a panel. The base plane is an alignment plane for all of the templates in the template set. For more information on the available base planes, see *Plate Template* (on page 148).



Base Control Line

Overrides the automatically generated base control line, and starts Sketch2D so that you can draw the base control line. The base control line is a basic alignment line for the templates within the set on the plate. It is also a reference for creating the template table.







Add Template

Starts Sketch2D so that you can draw a custom template location.



Bottom Line

Selects the action to perform from the list. The available options include the following:

-  **Pick Bottom Line** - Selects a bottom line to modify.
-  **Pick Point** - Selects a point on the base control line for creating a bottom line.
-  **Edit Bottom Line** - Starts Sketch2D so that you can modify the bottom line.
-  **Delete** - Deletes the manually edited bottom line.

Finish

Displays a dialog box asking if you want to accept or reject the changes. If you click **Accept**, the software displays a preview of the template set. This option creates the template using the parameters that you have defined.



Cancel

Clears the selection.



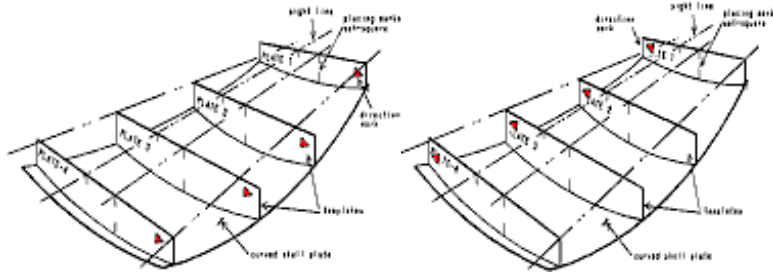
Accept

Accepts the selection.



Switch Upper/Lower

Reverses the seam location definition in the upper and lower direction.



Switch Fore/Aft

Reverses the seam location definition in the fore and aft direction.

BasePlane

Select the base plane type from the list. The available options include the following:

- **Most Planar Natural** - Generates a plane based on the planar portion of the plate surface. The planar portion should be at least 30 percent.
- **Average of Corners Plane** - Generates a plane based on the number of vertices of the plate surface. The system takes the normal from the plane defined by each combination of three vertices and calculates these normals based.
- **By System** - Generates the base plane based on system defaults.
- **User Defined** - Defines the base plane either by the Six Plane Control methods (for more information see *Plane Methods* (on page 38)) or the **By Three Points** method.
- **Parallel Axis** - Uses the coordinate axis which is most in line with the normal of the plate to define the base plane.
- **True Natural** - Generates the base plane from the normal at the center of the plate surface.
- **Lower Aft Corners** - Generates the base plane from the lower aft corners of the plate surface.
- **Upper Aft Corners** - Generates the base plane from the upper aft corners of the plate surface.
- **Lower Fore Corners** - Generates the base plane from the lower fore corners of the plate surface.
- **Upper Fore Corners** - Generates the base plane from the upper fore corners of the plate surface.
- **Mid Template Based** - Generates the base plane based on the plane at the mid-point of the plate.



By Three Points

Selects three vertex points to define the plane for the base plane. The software highlights the nearest vertex point to the cursor in red. When you select a vertex, it displays in yellow. This option is only available when **User Defined** is selected in the **BasePlane** list.

Side

Selects the side for the template.

- BaseSide

- OffsetSide
- MoldedSide
- AntiMoldedSide

Template Type

Selects the related template for its base plane and side. This option is only available if you select a plate that has defined templates defined.

Frame System

Selects a frame system to associate with the template. The list contains the frame systems from the catalog rule, as well as the frame system inherited from the parent assembly.

Process List

Selects a template process from the list. A process is a rule-based geometrical definition of a template based on particular manufacturing information. For example, template spacing is a process.

- Default_TemplateProcessPlate
- FrameEqualHeight_TemplatePlate
- Frame_TemplateProcessPlate
- Perpendicular_TemplateProcessPlate
- UserDefined_TemplateProcessPlate
- Even_TemplateProcessPlate
- CenterLine_TemplateProcessPlate
- StemStern_TemplateProcessPlate
- AftForward_TemplateProcessPlate
- PerpendicularXY_TemplateProcessPlate
- Box_TemplateProcessPlate
- UserDefinedBox_TemplateProcessPlate
- UserDefBoxEdges_TemplatesProcessPlate

Marking List










Select a process from the list.

- Default_TemplateMarkingPlate
- Box_TemplateMarkingPlate


What do you want to do?

- *Create a plate template set* (on page [155](#))
- *Modify a template set* (on page 155)
- *Copy and paste a template set* (on page 156)
- *Delete a template set* (on page 156)

Create a plate template set

1. Click **Template** .
2. On the **Template** ribbon, click **Plate Parts** .
3. From the **Template Type** list, select **Plate**.
4. Select the plate parts for which you want to create the template set.
TIP You can select the parts in the model, or from the **Workspace Explorer**.
5. Click **Base Plane**  if you want to change the balance type, template side, or choose a related template.
6. Click **Accept** .
7. Click **Base Control Line**  if you want to manually sketch the base control line. Using the available drawing commands in the 2D environment, draw the base control line.
8. Click **Add Template**  if you want to manually sketch template locations. Using the available drawing commands in the 2D environment, draw the templates.
9. Click **Accept**  to display a preview of the template set.
10. If necessary, click **Bottom Line**  to manually extend the bottom line of the template set.
11. Click **Finish** on the ribbon in the 2D environment.
12. Select the **Process** type.
13. Click **Properties** , and change the default **Process** and **Marking Set** values if needed.
14. Click **OK**.
15. Click **Finish**.


Modify a template set

1. Click **Select**  on the vertical toolbar.
2. Select the template set to modify.
TIP You can select the template set in the model or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the template set as needed.
NOTE You cannot change the parent plate when you modify the object.

Copy and paste a template set

1. Click **Select**  on the vertical toolbar.
2. Select the template set to copy.

TIPS

- You can select the template set in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Templates** to make the selection easier.
3. Click **Edit > Copy**.
 4. Click **Edit > Paste**.
 5. Select the part on which to paste the template set.
-  **TIP** You must select a detailed plate for the destination.
6. Click **OK** on the **Paste** dialog box.

Delete a template set



1. Click **Select**  on the vertical toolbar.
2. Select the template set to delete.
3. Click **Delete** .

Plate Template Properties Dialog Box

Specifies the properties for the template that you are editing.

See Also

Plate Template (on page 148)

General Tab (Template Properties Dialog Box) (on page 156)

Template Tab (Template Properties Dialog Box) (on page 158)

Process Tab for Plates (Template Properties Dialog Box) (on page 159)

Marking Tab (Template Properties Dialog Box) (on page 161)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Template Properties Dialog Box)

Name

Specifies the name of the template set. If you change the name, then the name rule changes automatically to **User Defined**.

Name Rule

Selects a name rule for the template set.

- Select **DefaultNameRule** to use the following syntax: <"T"><Block Name>-<Single Part Name>_<Template Index number>-<Workshare Location ID>. For example, TB0.1-7.1-4.1-2-1_1-1, where T denotes the template object, B0.1-7.1-4.1-2 is the block

name, 1 is the single part name, the 1 following the underscore is the template index number, and the final 1 is the Workshare Location ID.

- Select **User Defined** to manually type a name in the **Name** box.

Template Name Rule

Selects a name rule for the template.

Template Frame System

Specifies the frame system for the template set. The list contains the frame systems of the parent assembly systems, the frame systems provided by the rule, and the frame systems in the range of the template. By default, the template set inherits the frame system from the parent.

Surface Type

Displays the surface type used by the template. The list contains **Part Surface**, **Molded Surface**, and **Part Surface Base**. By default, **Part Surface** is used.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools > Recompute Objects**. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its

relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

See Also

Plate Template Properties Dialog Box (on page 156)

Template Tab (Template Properties Dialog Box)

This tab is available when you edit an existing template set.

Name

Displays the names of the templates in the selected template set. If you select a name, the corresponding template highlights in the model.

Index

Displays the index number assigned to each template. You can sort templates in each group by changing the index.

Group

Displays the group number assigned to each template. You can create groups of templates by changing the group number. By default, all templates along the primary direction are assigned to **Group1**, and all templates along the secondary direction (normal to the primary direction) are assigned to **Group2**. All other templates are assigned to other groups. There can be any number of groups.

IsReference

Indicates which templates are designated as the reference templates. Two templates, one each from the primary group and secondary groups, are defined as the reference templates by default. The intersection of the two reference templates is called the *reference point*.

Type

Indicates the type for each template in the template set.

- **System** - System generated template.
- **Sketched** - Template created by the **Add Sketched Templates** step.
- **Edited (System)** - System generated template modified in the **Bottom Line** step.
- **Edited (Sketched)** - Sketched template modified in the **Bottom Line** step.
- **Edited (Point)** - Template created by selecting a point on the base control line in the **Bottom Line** step.

See Also

Plate Template Properties Dialog Box (on page 156)

Process Tab for Plates (Template Properties Dialog Box)

Type

Specifies the type of template. Template types include:

- **frame**
- **center line**
- **perpendicular**
- **stern/stem**
- **perpendicular XY**
- **aft and forward**
- **user defined**
- **even**
- **box**
- **user defined box**
- **user defined box with edges**

Side

Specifies the side on which the template set is applied.

Orientation

Specifies the orientation for the template. Template orientations include:

- **along frame**
- **perpendicular**
- **normal to base plane**

Minimum Height

Specifies the minimum height of the template.

Maximum Height

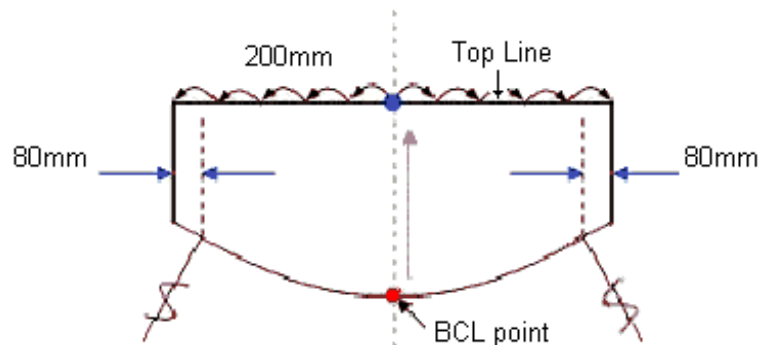
Specifies the maximum height of the template.

Position

Specifies the positions at which templates must be placed. This could be at **Frames**, at **Even Distance**, or at **Edges**.

Extension

Specifies the extension for the template.



Base Plane

Specifies the base plane options for the template. Base plane options include:

- **Normal to plate**
- **Parallel to plate**
- **By system**
- **User defined**

Direction

Specifies the direction of the template. Direction options include:

- **Longitudinal**
- **Transversal**
- **Waterline**

Template Service

Specifies the rule that computes the base plane, base control line, and template contours.

Type	Side	Min Height	Max Height	Orientation	Direction	Position Even	Position Frames	Extension	Base Plane
Frame	0	0	0	0	0 (Note 3)	0 (Note 1)	0	0	
Center Line	0	0	0	0	0	0 (Note 1)	0	0	
Perpendicular	0	0	0	0	0	0 (Note 1)	0	0	
Perpendicular XY	0	0	0	X	0	X	X	0	0 (Note 2)
Stern/ Stem	0	0	0	X	X	X	X	0	0 (Note 2)
User Defined	0	0	0	0	X	X	X	0	

Legend

0 - meaningful

X - not meaningful; ignored

NOTES

- If orientation is **AlongFrame**, **Position Even** should not be used and **PositionFrames** should be set to **PositionFrame**.
- The **BySystem** option is always applied because the **BasePlane** is generated by special logic in the code level.
- If **Type** is set to **Frame**, **Direction** should always be set to **Transversal (Y direction)**.

See Also

Plate Template Properties Dialog Box (on page 156)

Marking Tab (Template Properties Dialog Box)

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property will make that the value for all of the selected objects.

User Answers

Selects a template process type from the list. This list is the same as the **Process** list on the Template ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

You can modify the parameter values if necessary.

Rule-Based

If the value of the parameter is rule-based, a check box displays. You can clear the check box and then select a different parameter value if necessary.

Catalog Value

Displays the original catalog value for the process if you modified the value.

SideMark

For future use.

SeamMarks

Creates marks at seam locations at both ends of each template.

BaseCtlLineMark

Creates a mark on the top line of the template at some distance from the top base control line.

ShipDirectionMark

Creates a mark indicating the ship direction.

LabelMark

For future use.

FrameMarks

Creates marks of intersecting frames.

KnuckleMarks

Creates marks of intersecting knuckles with the template bottom line.

ReferenceMarks

Creates marks of intersecting reference curves with the template bottom line.

TemplateMarks

Creates marks of intersecting templates with the template.


CustomMarks

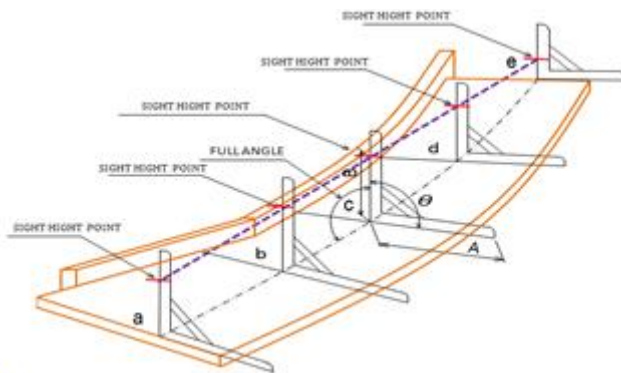
Creates any other marks based on rule.

See Also

Plate Template Properties Dialog Box (on page 156)

Profile Face Template

 Creates a set of templates at user-defined positions. The command generates the template outer contour shapes and reference point data that can be used for output in template table data.



Templates are used to check and control the actual curvature during and after the bending.

A template is a single object aligned with a respective marking line on a plate. A template set is a group of templates prepared for a single plate to check its curvature during and after the bending process.

In addition to specifying the shape and spacing for the templates, you can apply rule-based processes and marking sets to the templates, analogous to the processes and markings applied to plate parts.

NOTE If the software cannot compute the base plane or the base control line by rule, it displays a message box to warn you of the condition.

The software identifies the following template types:

Frame

A frame template set is a general way to make a template. This type of template is used in most cases. A frame template set consists of templates at each frame position on a selected plate. You can specify an interval value for defining the template positions, and you can add additional templates if needed.

Even

An even template set consists of templates at even distances along the base control line of the selected profile.

Aft and Forward

An aft and forward template set is similar to the frame template set, except that the location of the templates is determined by the aft and forward seam.

Perpendicular

A perpendicular template set specifies the templates at planes normal to the plate surface.

User Defined

A user defined template set lets you sketch the base control line and the template locations in the two-dimensional drafting environment. The template locations must intersect the base control line.

Profile Face Template Ribbon

Displays the controls used to add templates to the model.

Properties

Activates the **Template Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the template that you are about to place in the model. For more information, see *Plate Template Properties Dialog Box* (on page 156).

Profile Parts

Select the detailed profile part to which you want to add the template. You can select the part in a graphic view or in the **Workspace Explorer**.

Rule

Indicates if you are using a customized filter to select the part.

Template Type

Indicates the template type on which to place the template. The available options are:

- **Plate** - Places templates on plate parts.
- **Profile Face** - Places templates on profile faces.

Base Plane

Selects the base plane for the template. The base plane is an offset balance plane of a plate or a panel. The base plane is an alignment plane for all of the templates in the template set. For more information on the available base planes, see *Plate Template* (on page 148).

Base Control Line





Overrides the automatically generated base control line, and starts Sketch2D so that you can draw the base control line. The base control line is a basic alignment line for the templates within the set on the plate. It is also a reference for creating the template table.

Add Template

Starts Sketch2D so that you can draw a custom template location.

Bottom Line

Selects the action to perform from the list. The available options include the following:

-  **Pick Bottom Line** - Selects a bottom line to modify.
-  **Pick Point** - Selects a point on the base control line for creating a bottom line.
-  **Edit Bottom Line** - Starts Sketch2D so that you can modify the bottom line.
-  **Delete** - Deletes the manually edited bottom line.

Finish

Displays a dialog box asking if you want to accept or reject the changes. If you click **Accept**, the software displays a preview of the template set. This option creates the template using the parameters that you have defined.

Cancel

Clears the selection.

Accept

Accepts the selection.

Switch Upper/Lower

Reverses the seam location definition in the upper and lower direction.

Switch Fore/Aft

Reverses the seam location definition in the fore and aft direction.

BasePlane

Select the base plane type from the list. The available options include the following:

- **Most Planar Natural** - Generates a plane based on the planar portion of the plate surface. The planar portion should be at least 30 percent.
- **Average of Corners Plane** - Generates a plane based on the number of vertices of the plate surface. The system takes the normal from the plane defined by each combination of three vertices and calculates these normals based.
- **By System** - Generates the base plane based on system defaults.
- **User Defined** - Defines the base plane either by the Six Plane Control methods (for more information see *Plane Methods* (on page 38)) or the **By Three Points** method.
- **Parallel Axis** - Uses the coordinate axis which is most in line with the normal of the plate to define the base plane.

- **True Natural** - Generates the base plane from the normal at the center of the plate surface.



By Three Points

Selects three vertex points to define the plane for the base plane. The software highlights the nearest vertex point to the cursor in red. When you select a vertex, it displays in yellow. This option is only available when **User Defined** is selected in the **BasePlane** list.

Side

Selects the side for the template.

- WebLeft
- WebRight
- TopFlange
- BottomFlange

Template Type

Selects the related template for its base plane and side. This option is only available if you select a plate that has defined templates defined.

Frame System

Selects a frame system to associate with the template. The list contains the frame systems from the catalog rule, as well as the frame system inherited from the parent assembly.

Process List

Selects a template process from the list. A process is a rule-based geometrical definition of a template based on particular manufacturing information. For example, template spacing is a process.

- Default_TemplateProcessProfile
- Frame_TemplateProcessProfile
- Perpendicular_TemplateProcessProfile
- UserDefined_TemplateProcessProfile
- Even_TemplateProcessProfile
- AftForward_TemplateProcessProfile

Marking List










Select a process from the list.

- Default_TemplateMarkingProfile

What do you want to do?

- *Create a profile face template set* (on page 166)
 - *Modify a Template Set* (on page 155)
 - *Copy and Paste a Template Set* (on page 156)
 - *Delete a Template Set* (on page 156)
-

Create a profile face template set

1. Click **Template** .
2. On the **Template** ribbon, click **Profile Parts** .
3. From the **Template Type** list, select **Profile Face**.
4. Select the profile parts for which you want to create the template set.
TIP You can select the parts in the model, or from the **Workspace Explorer**.
5. Click **Base Plane**  if you want to change the balance type, template side, or choose a related template.
6. Click **Accept** .
7. Click **Base Control Line**  if you want to manually sketch the base control line. Using the available drawing commands in the 2D environment, draw the base control line.
8. Click **Add Template**  if you want to manually sketch template locations. Using the available drawing commands in the 2D environment, draw the templates.
9. Click **Accept**  to display a preview of the template set.
10. If necessary, click **Bottom Line**  to manually extend the bottom line of the template set.
11. Click **Finish** on the ribbon in the 2D environment.
12. Select the **Process** type.
13. Click **Properties** , and change the default **Process** and **Marking Set** values if needed.
14. Click **OK**.
15. Click **Finish**.

Profile Face Template Properties Dialog Box

Specifies the properties for the template that you are editing.

See Also

Profile Face Template (on page 162)

General Tab (Template Properties Dialog Box) (on page 156)

Template Tab (Template Properties Dialog Box) (on page 158)

Process Tab for Profile Faces (Template Properties Dialog Box) (on page 167)

Marking Tab (Template Properties Dialog Box) (on page 161)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

Process Tab for Profile Faces (Template Properties Dialog Box)

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a template process type from the list. This list is the same as the **Process** list on the **Template** ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

You can modify the parameter values if necessary.

Rule-Based

If the value of the parameter is rule-based, a check box displays. You can clear the check box and then select a different parameter value if necessary.

Catalog Value

Displays the original catalog value for the process if you modified the value.

Type

Specifies the type of the template. Template types include:

- **frame**
- **perpendicular**
- **user defined**
- **aft forward**
- **even**

Side

Specifies the side on which the template set is applied.

Orientation

Specifies the orientation for the template. Template orientations include:

- **along frame**
- **perpendicular**
- **normal to base plane**

Minimum Height

Specifies the minimum height of the template.

Maximum Height

Specifies the maximum height of the template.

Position

Specifies the positions at which templates must be placed. This could be at **Frames**, at **Even Distance**, or at **Edges**.

Extension

Specifies the extension for the template. Extensions include:

- **BothSidesFixed** - Creates a template on both sides of the base control line and extends that template on both sides with a fixed value.
- **BothSidesNo** - Creates a template on both sides of the base control line with no extension.
- **BothSidesOffset** - Creates a template on both sides with an extension based on the offset.
- **LowerSideFixed** - Creates a template with a fixed extension only on the lower side of the base control line.
- **LowerSideOffset** - Creates a template with an offset-based extension only on the lower side of the base control line.
- **LowerSideNo** - Creates a template with no extension only on the lower side of the base control line.
- **UpperSideFixed** - Creates a template with a fixed extension only on the upper side of the base control line.
- **UpperSideOffset** - Creates a template with an offset extension only on the upper side of the base control line.
- **UpperSideNo** - Creates a template with no extension only on the upper side of the base control line.

Service

Creates template profiles.

See Also

Profile Face Template Properties Dialog Box (on page 166)

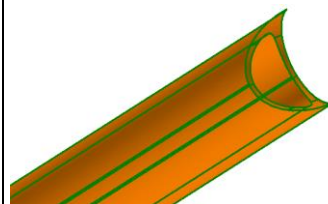
Tubular Member Template

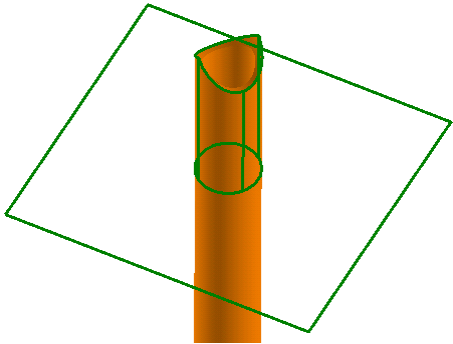
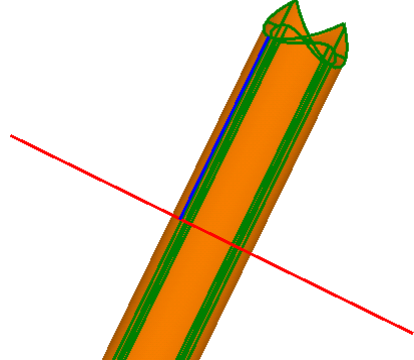
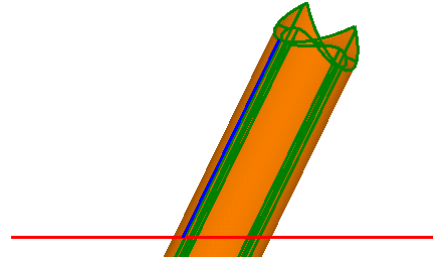
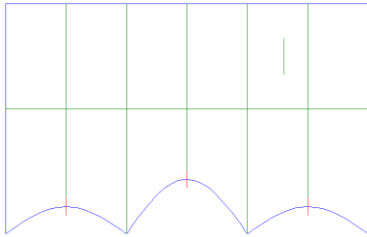


Creates and modifies templates to check the curvature of tube member end cuts.

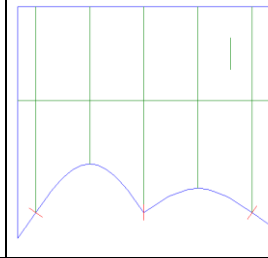
Tube template rules are delivered to the *[Product Folder]\SharedContent\Src\StructManufacturing\Rules\MfgTemplateSet\Definitions\Tube* folder.

Tube member with end cut.



<p>Tube template with base plane, side lines, and base control line on end cut.</p>	
<p>Base plane perpendicular to axis.</p>	
<p>Base plane in global direction.</p>	
<p>Template output with short distance base control line and ExtnRad90.</p>	

Template output with base control line at the model origin and ExtnLinear100.



Tubular Member Template Ribbon

Displays the controls used to add tube templates to the model.



Properties

Activates the **Template Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the template that you are about to place in the model. For more information, see *Tube and Edge Templates Properties Dialog Box* (on page 172).



Tubular Member Parts

Select the tubular member part to which you want to add the template. You can select the part in a graphic view or in the **Workspace Explorer**.



Base Plane

Select the base plane for the template. The base plane is an offset balance plane based on the tube axis. The base plane can be perpendicular to the tube axis, or it can be along one of the global axes.

Base Plane

- By System
- NormalToAxis
- Global
- UserDefined

Side

- Outer
- Inner

Side End

- Base
- Offset

Process List

- Default_TemplateProcessTube
- ShortDistBCL_TemplateProcessTube
- LongDistBCL_TemplateProcessTube

Marking List

- Default_TemplateMarkingTube

**Base Control Line**

Currently, this step is not supported. It overrides the automatically generated base control line and starts Sketch2D so that you can draw the base control line. The base control line controls the position on the 3D tube template where the tube is opened to unfold. The tube opens at the opposite position to the base control line and the 2D tube template output is generated by unfolding the 3D tube template.

Finish

Displays a dialog box asking if you want to accept or reject the changes. If you click **Accept**, the software displays a preview of the template set. This action creates the template using the parameters that you have defined.

**Cancel**

Clears the selection.

**Accept**

Accepts the selection.

Process List

Select a template process from the list. A process is a rule-based geometrical definition of a template based on particular manufacturing information. For example, template spacing is a process.

- Default_TemplateProcessTube
- ShortDistBCL_TemplateProcessTube
- LongDistBCL_TemplateProcessTube

Marking List






Select a process from the list.

- Default_TemplateMarkingTube


What do you want to do?

- *Create a tube template* (on page [171](#))
 - *Modify a tube template* (on page [172](#))
 - *Delete a tube template* (on page [172](#))
-



Create a tube template

1. Click **Tube Template** .
2. On the **Tube Template** ribbon, click **Tubular Member Parts** .
3. Select the tube parts for which you want to create the template set.
TIP You can select the parts in the model or from the **Workspace Explorer**.
4. Click **Base Plane**  if you want to change the balance type, template side, or choose a related template.
5. Click **Accept** .
6. Select the **Process** type.
7. Click **Properties** , and change the default **Process** and **Marking Set** values, if needed.
8. Click **OK**.
9. Click **Finish**.

Modify a tube template

1. Click **Select**  on the vertical toolbar.
2. Select the tube template to modify.
TIP You can select the template in the model or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the tube template as needed.
NOTE You cannot change the parent plate when you modify the object.

Delete a tube template

1. Click **Select**  on the vertical toolbar.
2. Select the tube template to delete.
3. Click **Delete** .

Tube and Edge Templates Properties Dialog Box

Specifies the properties for the template that you are editing.

See Also

Tubular Member Template (on page 168)

General Tab (Tube and Edge Template Properties Dialog Box) (on page 173)

Template Tab (Tube and Edge Template Properties Dialog Box) (on page 174)

Process Tab (Tube and Edge Template Properties Dialog Box) (on page 175)

Marking Tab (Tube and Edge Template Properties Dialog Box) (on page 177)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Tube and Edge Template Properties Dialog Box)

Name

Specifies the name of the template set. If you change the name, then the name rule changes automatically to **User Defined**.

Name Rule

Selects a name rule for the template set.

- Select **DefaultNameRule** to use the following syntax: <"T"><Block Name>-<Single Part Name>_<Template Index number>-<Workshare Location ID>. For example, TB0.1-7.1-4.1-2-1_1-1, where T denotes the template object, B0.1-7.1-4.1-2 is the block name, 1 is the single part name, the 1 following the underscore is the template index number, and the final 1 is the Workshare Location ID.
- Select **User Defined** to manually type a name in the **Name** box.

Template Name Rule

Selects a name rule for the template.

Template Frame System

Specifies the frame system for the template set. The list contains the frame systems of the parent assembly systems, the frame systems provided by the rule, and the frame systems in the range of the template. By default, the template set inherits the frame system from the parent.

Surface Type

Displays the surface type used by the template. The list contains **Part Surface**, **Molded Surface**, and **Part Surface Base**. By default, **Part Surface** is used.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools > Recompute Objects**. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

See Also

Tube and Edge Templates Properties Dialog Box (on page 172)

Template Tab (Tube and Edge Template Properties Dialog Box)

Because tube template sets contains a single template, only one template displays in this tab.

Name

Displays the names of the templates in the selected template set. If you select a name, the corresponding template highlights in the model.

Index

Displays the index number assigned to each template. You can sort templates in each group by changing the index.

Group

Displays the group number assigned to each template. You can create groups of templates by changing the group number. By default, all templates along the primary direction are assigned to **Group1**, and all templates along the secondary direction (normal to the primary direction) are assigned to **Group2**. All other templates are assigned to other groups. There can be any number of groups.

IsReference

Indicates which templates are designated as the reference templates. Two templates, one each from the primary and secondary groups, are defined as the reference templates by default. The intersection of the two reference templates is called the *reference point*.

Type

Indicates the type for each template in the template set.

- **System** - System generated template.

See Also

Tube and Edge Templates Properties Dialog Box (on page 172)

Process Tab (Tube and Edge Template Properties Dialog Box)

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a template process type from the list. This list is the same as the **Process** list on the **Template** ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

You can modify the parameter values if necessary.

Rule-Based

If the value of the parameter is rule-based, a check box displays. You can clear the check box and then select a different parameter value if necessary.

Catalog Value

Displays the original catalog value for the process if you modified the value.

Minimum Height

Specifies the minimum height of the base plane from the end cut. Available options include:

- **Fixed**
- **Height_50**
- **Height_100**
- **Height_150**
- **Height_200**

Maximum Height

Specifies the maximum height of the base plane from the end cut.

Extension

Specifies the extension to apply on the bottom line. Available options include:

- **ExtnRad90**
- **ExtnLinear100**
- **ExtnAlongEdge50**
- **Ignore**

Side

Specifies the side on which the template set is applied.

- **Outer**
- **Inner**

SideEnd

Specifies the tube end on which the template set is applied. Available options include:

- **Base**
- **Offset**

Type

Specifies the template type based on the base control line position. Template types include:

- **ShortDistance BCL**
- **LongDistanceBCL**
- **ModelOrigin**
- **Global X Max**
- **Global X Min**
- **Global Y Max**
- **Global Y Min**
- **Global Z Max**
- **Global Z Min**

Base Plane

Specifies the base plane option. Base plane options include:

- **BySystem**
- **NormalToAxis**
- **Global**
- **User defined**

Template Service

Specifies the rule that computes the base plane, base control line, and template contours.

UserDefinedValues

Specifies the rule the returns any user-defined values.

TemplateName

Specifies the rule the computes the template name.

Bevel

Specifies the rule the computes the bevel.

See Also

Tube and Edge Templates Properties Dialog Box (on page 172)

Marking Tab (Tube and Edge Template Properties Dialog Box)

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a template process type from the list. This list is the same as the **Process** list on the Template ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

You can modify the parameter values if necessary.

Rule-Based

If the value of the parameter is rule-based, a check box displays. You can clear the check box and then select a different parameter value if necessary.

Catalog Value

If you modified the value, the original catalog value for the process displays.

BaseCtlLineMark

Creates a mark on the top line of the template at some distance from the top base control line.

FittingMark

Creates fitting marks. This option only applies to tube templates.

FrameMarks

Creates marks of intersecting frames.

QuarterLineMarks

Creates quarter line marks. This option only applies to tube templates.

SeamMarks

Creates seam marks.

ShipDirectionMark

Creates ship direction marks.

CustomMark

Creates custom marks.

See Also

Tube and Edge Templates Properties Dialog Box (on page 172)

Edge Template



Creates and modifies templates to check the curvature of edges.

Edge template rules are delivered to the *[Product Folder]\SharedContent\Src\StructManufacturing\Rules\MfgTemplateset\Definitions\Edge* folder.

Edge Template Ribbon

Displays the controls used to add edge templates to the model.



Properties

Activates the **Template Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the template that you are about to place in the model. For more information, see *Tube and Edge Templates Properties Dialog Box* (on page 172).



Edge Parts

Select the edge part to which to add the template. You can select the part in a graphic view or in the **Workspace Explorer**.

Finish

Displays a dialog box asking if you want to accept or reject the changes. If you click **Accept**, the software displays a preview of the template set. This action creates the template using the parameters that you have defined.



Cancel

Clears the selection.



Accept

Accepts the selection.

Rule

Indicates if you are using a customized filter to select the part.

Side

Selects the side for the template.

- WebLeft
- WebRight
- Top
- Bottom

Offset

Select the offset for the template.

- No_Offset
- Offset_10
- Offset_20
- Full_Offset

Process List

Select a template process from the list. A process is a rule-based geometrical definition of a template based on particular manufacturing information. For example, template spacing is a process.

- Default_TemplateProcessEdge

Marking List




Select a process from the list.

- Default_TemplateMarkingEdge


What do you want to do?

- *Create an edge template* (on page [180](#))
 - *Modify an edge template* (on page 180)
 - *Delete an edge template* (on page 180)
-



Create an edge template

1. Click **Edge Template** .
2. On the **Edge Template** ribbon, click **Edge Parts** .
3. Select the edge parts for which you want to create the template set.
TIP You can select the parts in the model or from the **Workspace Explorer**.
4. Select the side for the template from the **Side** list.
5. Select the type of offset to use from the **Offset** list.
6. Select the process type to use from the **Process List** list.
7. Click **Properties** , and change the default **Process** and **Marking Set** values, if needed.
8. Click **OK**.
9. Click **Finish**.

Modify an edge template

1. Click **Select**  on the vertical toolbar.
2. Select the edge template to modify.
TIP You can select the template in the model or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the edge template as needed.
NOTE You cannot change the parent plate when you modify the object.

Delete an edge template

1. Click **Select**  on the vertical toolbar.
2. Select the edge template to delete.
3. Click **Delete** .

SECTION 7

Pin Jig



Creates a set of pin jigs at positions defined in the work center. The work center is defined in the Planning task. This command also allows you to add rows, columns, and individual pin locations.



A pin jig is a manufacturing aid consisting of a height-adjustable pedestal that supports a part such as a plate, assembly, or block.

NOTE You can place multiple pin jigs on a plate, assembly, or block.

When you place pin jigs, you must specify the base plane of the supported parts. The base plane is a plane that represents the workshop floor. The software calculates the height of the pins as the distance between the base plane and the part, taking into account the used pin head. The spacing between the jig grid lines is derived from the work center information that has been assigned to the assembly to which the parts belong.

Pin Jig Ribbon

Displays the controls used to define a pin jig.



Properties

Activates the **Pin Jig Properties** dialog box, which you use to view and modify the properties of the pin jig that you are about to place in the model. For more information, see *Pin Jig Properties Dialog Box* (on page 194).



Pin Jig Parent

Select the detailed plate part or assembly that is the parent of the pin jig.

NOTE A detailed plate part or assembly can have multiple pin jigs.



Select

Manually select the supported plate parts. This button only displays if you select an assembly.



Select Supported Plate Parts

Displays the **Supported Plate List** dialog box. For more information, see *Supported Plate List Dialog Box* (on page 187).



Accept

Accepts the selected supported objects.



Cancel

Cancels the object selection.

Select Boundaries

Define the outer contour seams and butts for the pin jig, which is required for reporting. The aft, lower, fore, and upper seam selections are also used to rename the seams on the drawings.


Boundary Steps

Aft

Specifies the aft boundary. If the software can compute the aft boundary, that is the default.

Lower

Specifies the lower boundary. If the software can compute the lower boundary, that is the default.

 **NOTE** The aft and lower boundaries must have a common point. This point determines the origin point for the plate parts on the pin jig.

Fore

Specifies the fore boundary. If the software can compute the fore boundary, that is the default.

Upper

Specifies the upper boundary. If the software can compute the upper boundary, that is the default.

NOTES

- You can select multiple boundary objects within each seam definition. For example, the aft boundary can be comprised of several seams, plate boundaries, and so on that are linked together.
- You can select seams, or any object that would form a plate boundary, to define the contour boundaries.

Aft Boundary Name

Displays the name assigned to the aft boundary. You can change the default to meet your needs.

Lower Boundary Name

Displays the name assigned to the lower boundary. You can change the default to meet your needs.

Fore Boundary Name

Displays the name assigned to the fore boundary. You can change the default to meet your needs.

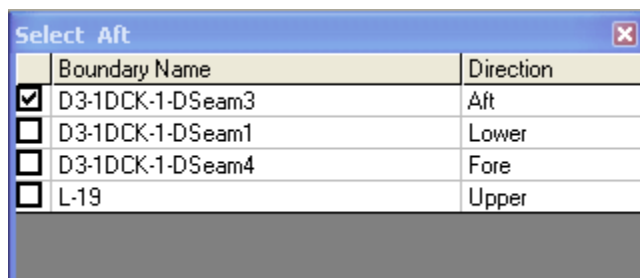
Upper Boundary Name

Displays the name assigned to the upper boundary. You can change the default to meet your needs.

Display List

Indicates whether the **Boundary Steps** dialog box displays.

The **Boundary Steps** dialog box displays a list of boundaries for the selected plates. For a regular plate with four boundaries, this dialog box displays the list of boundaries and the corresponding directions. For irregular plates, such as plates forming an L shape, the dialog box displays the boundaries but does not assign directions. You can specify the boundary by selecting the corresponding checkbox. When you select the boundary, it highlights in the model. A single boundary can be assigned to multiple directions. Multiple boundaries can be assigned to a single direction.



Base Plane

Specify the base plane for the pin jig. The base plane defines the jig floor.

Balance Method

Specifies the balance method for the pin jig. For some methods, you can select the inputs from the model by selecting the **User Inputs** option.





Direction Toggle

Reverses the direction of the base plane.

User Inputs

Indicates that you select inputs for the **Balance Method** from the model. Valid inputs include:

-  **Reference lines** - Seams, knuckles, reference curves, and marking lines that pass over the selected plates.
-  **Select Points** - Plate corners, end points of seams, knuckles, reference curves, and marking lines.

NOTE If the balance method is set to **User Defined**, the **Plane Definition Control** buttons display.



Coincident Plane

Specifies that you want the plate part placed on the reference plane.



By Plane or Offset Plane

Places the plate system a specified distance from the reference plane. If you choose this option, you must define the offset distance. An offset distance of zero defines a coincident plane.



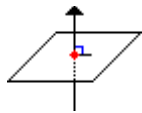
Angle from plane

Places the plate part at a specified angle or slope to the reference plane. If you choose this option, you must define an axis of rotation and the angle or slope.



Plane by Point and Vector

Specifies the reference plane by defining a vector perpendicular to the reference plane. A third point defines the reference plane position along the vector.






Plane by Vectors Normal

Specifies the reference plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.



Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane**  maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit > Paste > Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane**  is not visible. You must first click **Select Bracket Supports**  to allow the command to determine the applicable options.



Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves. The **Lock** option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance for the plate part from the selected reference plane. You can specify the offset dynamically in graphics or by typing the distance. The **Offset** option is only available when you use the **Offset from a Plane** option.

Angle

Specify the angle at which to place the plate part relative to the reference plane. You have to define the axis of rotation using two points before you can define the angle. The **Angle** option is only available when you use the **Angle from plane** option.

Step

Specifies the angle step. The step is incremented by this value when the cursor is moved in the graphic view. The **Step** option is only available when you use the **Angle from plane** option.



Select Vector

Select or define a vector normal to the needed plane. This option is only available when you use the **Plane by Point and Vector** option.



Define Point

Specify the point along the vector at which to place the plane. This option is only available when you use the **Plane by Point and Vector** option.



Define Point 1

Specify the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

 **Define Point 2**

Specify the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

 **Define Point 3**

Specify the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

 **Select Plane**

Select the plane to which your reference plane is to be normal. This option is only available when you are using the **Plane by Vectors Normal** option.

 **Select First Point**

Specify the location of the first vector point. Your reference plane is parallel to this vector. This option is only available when you are using the **Plane by Vectors Normal** option.

 **Select Second Point**

Specify the location of the second vector point. Your reference plane is parallel to this vector. This option is only available when you are using the **Plane by Vectors Normal** option.

 **Remarking Lines**

Displays the **Remarking Lines** dialog box from which you can modify existing remarking lines and add new remarking lines. You can also view and modify the attribute values of the remarking lines. For more information, see *Remarking Lines Dialog Box* (on page 188).

 **Jig Setup**



Activates the **Pin Jig 3D View** dialog box. For more information, see *Pin Jig 3D View Dialog Box* (on page 189).

Finish

Creates the pin jig.

Pin Alignment

Specifies the alignment of the pins on the jig floor. This option is available when **Base Plane**

 is selected during placement of a new pin jig. It is also available when **Jig Setup**  is selected during creation and modification of an existing pin jig. The available options are:

- **Default** - Pins are aligned based on the tightest box bounding the supported plates.
- **Longest Edge** - Pins are aligned based on the longest contour. The rows or columns of pins are parallel to the line joining the end points of the longest contour.
- **Global** - Pins are aligned based on the global coordinate system. The rows or columns of pins are parallel to the global axes.
- **Centerline** - Pins are aligned based on the centerline. A row of pins is placed on the centerline. This option is only available when the selected plates cross the centerline of the hull.

Minimum Pin Height

Select the minimum height of the pins from the list. This option is set automatically if the **Pin Height Rule** box is set to **By Rule**.

Pin Height Rule

Select the rule from the list.








Equipment

Select the equipment to use from the list. This information comes from the work centers assigned to the assembly to which the supported parts belong.


What do you want to do?

- *Create a pin jig* (on page 186)
- *Modify a pin jig* (on page 187)
- *Copy and paste a pin jig* (on page 187)
- *Delete a pin jig* (on page 187)

Create a pin jig


1. Click **Pin Jig** .
2. On the **Pin Jig** ribbon, click **Plate Parts** .
3. Select the plate part or assembly for which you want to create the pin jig.
TIP You can select the part in the model, or from the **Workspace Explorer**.
4. If creation of a pin jig has previously failed on the selected plate part, click **Define Seams and Butts** , and select the butts and seams.
5. Select the balance method that you want to use from the **Balance Method** list.
6. If you select the **User Defined** balance method, click **Base Plane**  and specify a plane to represent the shop floor for the jig.
 If you select the **Most Planar Natural** balance method, specify the percentage of planarity for the supported plates to define the base plane.
7. For some of the balance methods, you can specify the inputs by selecting the **User Inputs** option.
 - a. **Select Reference Line**  - Select a seam, knuckle, reference curve, or marking line that passes over the selected plates.
 - b. **Select Points**  - Select the points necessary depending on the **Balance Method** selected. These points can be plate corners, end points of seams or knuckles, reference curves, or marking lines.
8. If necessary, click **Remarking Lines** to view and modify the remarking lines.
9. Click **Modify Jig Setup** .
10. Using the commands in the **Pin Jig 3D View**, modify the jig layout if needed.
11. Click **Finish**.

Modify a pin jig


1. Click **Select**  on the vertical toolbar.
2. Select **Pin Jig Part** in the **Locate Filter** box.
3. Select the part or pin jig.
TIP You can select the part or pin jig in the model, or from the **Workspace Explorer**.
4. Using the ribbon controls, edit the pin jig as needed. For more information, see *Pin Jig* (on page 181).

NOTE You cannot change the parent plate when you modify the object.

Copy and paste a pin jig

1. Click **Select**  on the vertical toolbar.
2. Select the pin jig to copy.
TIPS
 - The pin jig must be the child of a plate. It cannot be the child of an assembly.
 - You can select the pin jig in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Pin Jigs** to make the selection easier.
3. Click **Edit > Copy**.
4. Click **Edit > Paste**.
5. Select the plate on which to paste the pin jig.
TIP You must select a detailed plate for the destination.
6. Click **OK** on the **Paste** dialog box.

Delete a pin jig

1. Click **Select**  on the vertical toolbar.
2. Select the pin jig to delete.
3. Click **Delete** .

Supported Plate List Dialog Box

Specifies all of the plate parts within the selected assembly so that you can specify which plate parts are supported by the pin jig.

Current List

Displays the currently selected plate parts.

Potential List

Displays all of the plate parts within the selected assembly. By default this list contains all of the shell plates under the selected assembly.

Add

Moves the selected plate parts from the **Potential List** to the **Current List**.

Remove

Moves the selected plate parts from the **Current List** to the **Potential List**.

Clear All

Moves all plate parts from the **Current List** to the **Potential List**.

Select Mfg Parts Dialog Box

Lists all of the manufacturing parts, such as marking lines and pin jigs, associated with the selected detailed part so that you can select the object to review or modify. Use the selection filter to limit the types of objects included in the list.

Remarking Lines Dialog Box

Tree View

Displays the remarking lines associated with the pin jig. When you expand a node in the tree view, the entities represented by the remarking line display. There may also be a node called **SKIPPED ENTITIES** if entities are skipped for remarking.

The following properties display when you select a remarking line from the tree view:

Name

Displays the name of the selected marking line.

Direction

Displays the direction of the remarking line. Available options include **Auto**, **Horizontal**, and **Vertical**.

Remarking Type

Displays the remarking type associated with the remarking line. This option is not editable.

Intersection Point

Indicates whether the remarking line is used for the creation of intersection points.

Sorted Index

Displays the index value associated with the remarking line.

The following properties display when you select the entity from the tree view

Name

Displays the name of the selected entity.

Remarking Line Name

Displays the parent of the selected entity. You can change the parent or select **New Remarking Line** to assign the entity to a remarking line that does not yet exist. This option is

also available for skipped entities

Filter Objects

Displays all remarking lines with a particular attribute value. Click **Filter Objects**, and then select the attribute value by which to filter the objects.

Merge

Combines the selected remarking lines in the tree control.

Unmerge

Separates the selected remarking lines in the tree control.

Select All

Selects all of the remarking lines in the tree control, and highlights all of the remarking lines in the model view.

Pin Jig 3D View Dialog Box

Specifies the graphic layout for the pin jig that you are editing.

 **Select**

Selects pins to display their attributes. The type of object that you can select is determined by the value in the **Object Type** box.

View Type

Specifies the view orientation for the pin jig.

 **Rotate View**

Rotates the view 90 degrees in the clockwise direction.

 **Move**

Moves the entire jig set relative to the pin jig layout.

 **Move From**

Defines the starting point for the move.

 **Move To**

Defines the ending point for the move.

Horizontal Offset

Specifies a horizontal offset value.

Vertical Offset

Specifies a vertical offset value.

Move

Moves the jig set using the parameters that you have specified.

 **Rotate**

Rotates the entire jig set about the specified point.

Rotation Method

Specifies the method for rotating the PinJig. The available options are:

- **Angle** - Applies rotation by an angle about a point. View type decides the axis of rotation.
- **Pin Heights** - Applies rotation about a point such that the reference pin gets the specified height after rotation. This option is not available for side-view rotation.
- **Intersection Point Heights** - Applies rotation about a point such that the reference intersection point gets the specified height after rotation. This option is not available for side-view rotation.



Reference Point

Specifies the reference pin whose height is updated on rotation. This option is only available if the **Rotation Method** is set to **Pin Heights**.



Reference Point

Specifies the reference intersection point whose height is updated on rotation. This option is only available if the **Rotation Method** is set to **Intersection Point Heights**.



Reference Intersection Point

Specify the intersection point to use as the reference point in the girth length computation for all the intersection points in the pin jig. This point has no relationship with the rotate or move commands.



Add Pin

Add pins one at a time to the jig set. These pins are added at horizontal and vertical offset from an existing pin.



Add Pins

Add a row of pins to the jig set. These pins are added as offset from the row of an existing pin.

Add

Adds the pin to the jig set using the parameters that you have specified.

Direction

Select a second pin to specify the direction from the selected pin in which you want to place the row or column of pins.

Offset

Specify a distance from the selected pin for the row of pins.



Delete Pin

Deletes a manually added pin.


Object Type

Determines which objects are selectable.

Pin Alignment

Specifies the alignment of the pins on the jig floor. This option is available when **Base Plane**






is selected during placement of a new pin jig. It is also available when **Jig Setup**  is

selected during creation and modification of an existing pin jig. The available options are:

- **Default** - Pins are aligned based on the tightest box bounding the supported plates.
- **Longest Edge** - Pins are aligned based on the longest contour. The rows or columns of pins are parallel to the line joining the end points of the longest contour.
- **Global** - Pins are aligned based on the global coordinate system. The rows or columns of pins are parallel to the global axes.
- **Centerline** - Pins are aligned based on the centerline. A row of pins is placed on the centerline. This option is only available when the selected plates cross the centerline of the hull.

Display Text

Indicates that the pin height text displays. The available options are:

-  **Pin heights** - Displays the heights of all the pins in the view.
-  **Intersection point heights** - Displays the heights of all the intersection points in the view.
-  **Remarking girth lengths** - Displays the lengths of all the remarking girths in the view.

Attribute Viewer

Displays information about the selected object. The information displayed depends upon the type of the selected object.



Select Pivot Point

Specifies the point about which to rotate the pin jig. This point can be any point in the setup view. If the **Rotation Method** box is set to **Angle**, this point defaults to the (H0, V0) pin. There is no default for the other rotation methods.

Rotate

Rotates the jig set around the specified pin, using the parameters that you have specified.

Angle

Specify the rotation angle.

Reference Height

Specify the height of the reference object after rotation.

Pin

Name

The pin name in row, column format.

Row

The row to which the pin belongs.

Column

The column to which the pin belongs.

Height

For a pin, this is the height of the pin. For an intersection point, this is the height of the intersection point from the floor.

Added

Indicates whether or not the pin was manually added.

Corner

Indicates whether or not the pin is a corner pin.

Horizontal Marking Distance

Indicates the horizontal distance between the pin and the nearest remarking line.

Vertical Marking Distance

Indicates the vertical distance between the pin and the nearest remarking line.

Nearest Corner Point

Indicates the nearest corner point from the selected pin.

Horizontal Distance to Nearest Corner Point

Indicates the horizontal distance between the pin and the nearest corner point.

Vertical Distance to Nearest Corner Point

Indicates the vertical distance between the pin and the nearest corner point.

Minimum Pin Information**Row**

The row to which the minimum pin belongs.

Column

The column to which the minimum pin belongs.

Height

The height of the minimum pin.

Added

Indicates whether or not the minimum pin was manually added.

Intersection Point**Name**

The type of the intersection point, such as a corner point.

Base Position

The position on the jig floor.

Upper Position

The position on the supported plates.

Cusp Angle

The change in direction at the cusp point.

Horizontal Remarking Line

The name of the horizontal remarking line through which the point passes.

Vertical Remarking Line

The name of the vertical remarking line through which the point passes

NOTE If there are multiple remarking lines that overlap one another, the software chooses the longest remarking line to participate in creating intersection points with other crisscrossing remarking lines. If the overlapping remarking lines are identical, the software chooses based on the priority set by the pin jig rules. By default, this order is contour lines, then frames, and then seams.

Closest Pin Information**Name**

The name of the closest pin in row, column format.

Row

The row to which the closest pin belongs.

Column

The column to which the closest pin belongs.

Horizontal Distance from Pin

The horizontal distance between the closest pin and the intersection point.

Vertical Distance from Pin

The vertical distance between the closest pin and the intersection point.

Remarking Line**Name**

The name of the remarking line.

Girth Length

The length through the remarking line.

Straight Length

The distance between the start and end points of the remarking line.

Start Point Information**Name**

The name of the starting point of the remarking line.

Position

The position of the starting point on the supported plates.

End Point Information


Name

The name of the ending point of the remarking line.

Position

The position of the ending point on the supported plates.

Remarking Girth

 **NOTE** The remarking girth is the portion of a remarking line which lies between two intersection points.

Remarking Line

Select the remarking line.

First Intersection Point

Select the first intersection point on the selected remarking line.

Second Intersection Point

Select the second intersection point on the selected remarking line.

- **Girth Length** - The portion of the remarking line selected.
- **Straight Length** - The distance between the start and end points of this portion of the remarking line.

Remarking Line

Name

The name of the remarking line.

1st Intersection

Position

The location of the first point of the remarking line segment.

2nd Intersection

Position

The location of the second point of the remarking line segment.

See Also

Pin Jig (on page 181)

Marking (on page 65)

Pin Jig Properties Dialog Box

Specifies the properties for the pin jig that you are editing.

See Also

General Tab (Pin Jig Properties Dialog Box) (on page 195)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Pin Jig Properties Dialog Box)

Standard

Name

Specifies the name of the pin jig set. If you change the name, then the name rule changes automatically to **User Defined**.

Name Rule

Select a name rule for the jig set.

- Select **DefaultNameRule** to use the following syntax for a plate system: <"PJ">_<Parent Name>-<Workshare Location ID>-<Index number>. If the parent is an assembly system, the following syntax is used: <"AJ">_<Parent Name>-<Workshare Location ID>-<Index number>. For example, AJ_A10-1-103, where AJ represents the type of pin jig being used, A10 is the assembly parent name, 1 is the workshare location ID, and 103 is a reference number.
- Select **User Defined** to type a name in the **Name** box.

Frame System

Specifies the frame system for the pin jig set. The list contains the frame systems of the parent assembly systems, the frame systems provided by the rule, and the frame systems in the range of the pin jig.



Indicates that the value is inherited from the parent. By default, the pin jig set inherits the frame system from the parent.

Pin Origin Position

Specifies the position of the pin origin. This property is available in both the creation and modification modes of the command. The possible values are:

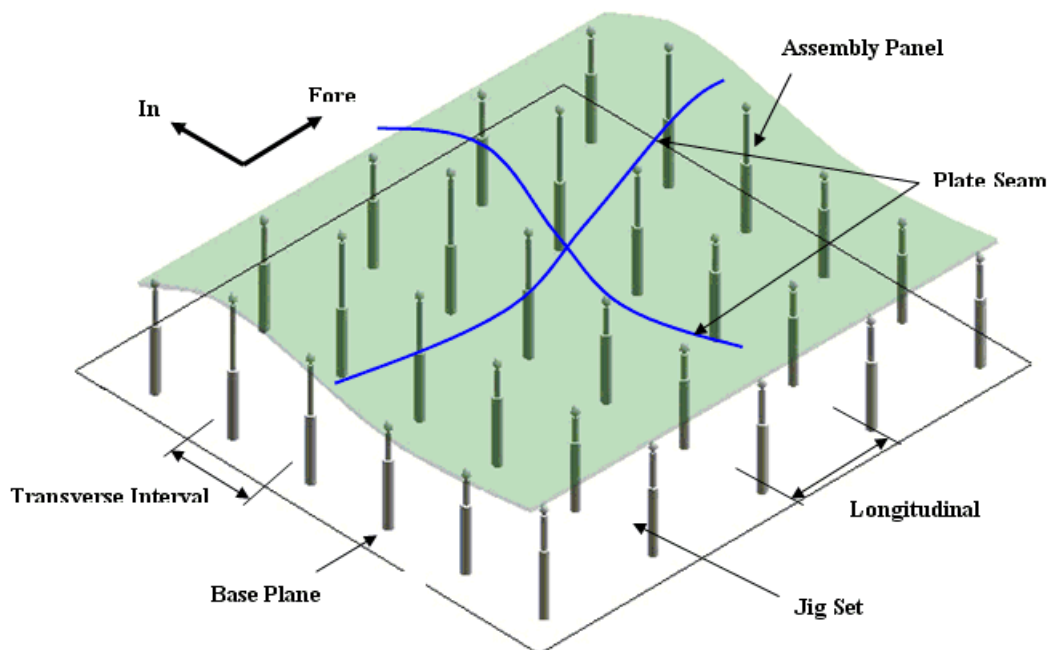
- **LowerLeft**
- **LowerRight**
- **UpperLeft**
- **UpperRight**

Balance Method

Specifies the balance type assigned to the pin jig. This is a read-only box.

Jig Floor

Specifies the jig floor associated with the pin jig set. This is a read-only box.



Longitudinal Size

Specifies the longitudinal length of the entire jig floor. You can use this value to verify the assembly range for the selected pin jig structural assembly or part.

Transversal Size

Specifies the transversal width of the entire jig floor. You can use this value to verify the assembly range for the selected pin jig structural assembly or part.

Longitudinal Range

Specifies the maximum longitudinal length of an assembly placed on the jig floor. You can use this value to verify the assembly range for the selected pin jig structural assembly or part.

Transversal Range

Specifies the maximum transversal width of an assembly placed on the jig floor. You can use this value to verify the assembly range for the selected pin jig structural assembly or part.

Longitudinal Interval

Specifies the longitudinal interval between pins. The software uses this value during the pin jig computation.

Transversal Interval

Specifies the transversal interval between pins. The software uses this value during the pin jig computation.

Longitudinal Naming

Specifies the naming convention used for longitudinal pin positions.

- **0** Indicates numeric naming.
- **1** Indicates alphabetic naming.

Transversal Naming

Specifies the naming convention used for transversal pin positions.

- **0** Indicates numeric naming.
- **1** Indicates alphabetic naming.

Pitch Floor Margin

Specifies the maximum allowable distance between the assembly and the jig contour..

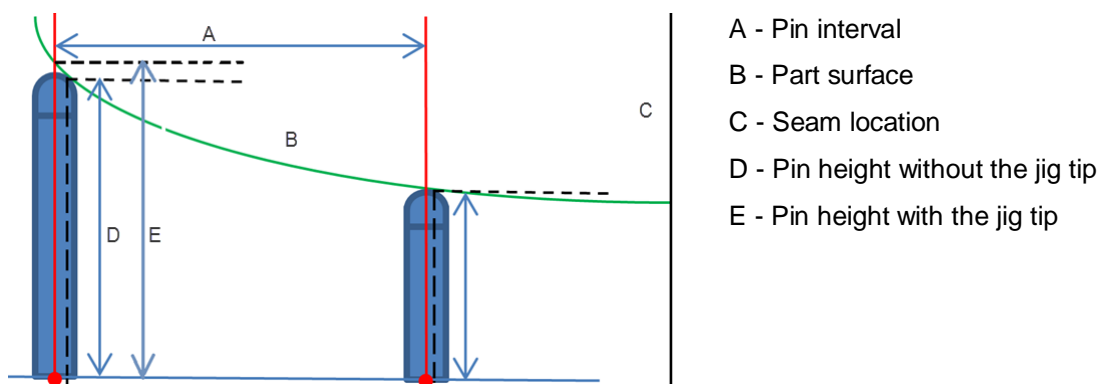
Pin Control

Specifies the method used to set the pins into position.

- **0** Indicates manual.
- **1** Indicates automatic or computer-controlled.

Jig Tip Type

Specifies the tip type (such as corn, half sphere, or diamond) for the pins. The software uses this value to calculate the real pin height. This height includes the gap between the pin head and the part surface.



Pin Minimum Height

Specifies the minimum height of the pin jig set. The software uses this value during the pin jig computation.

Pin Maximum Height

Specifies the maximum height of the pin jig set. The software uses this value during the pin jig computation.

Seam Distance

Specifies the minimum distance between pins and seam lines. This value is of significance to those pins that are close to an outer contour or inner seam.

Pin Jig Type

Specifies the pin jig type set by the software. This is a read-only box.

Pin Jig Purpose

Specifies the purpose of the pin jig.

Marking Settings

Apply Rule

Specifies whether the display of remarking lines is based on a rule, or is user defined. If this option is selected, all of the rules display as read-only. If this option is not selected, the rules are editable.

 **NOTE** This check box is only available during the creation of the pin jig.

Pin Jig Remarking Rule Set

Specifies the rule set that the software uses to get the geometry for remarking lines of each type.

Default - Identical to **Based on Physical Connection**. The options is available so that legacy models continue to work.

Based on Logical Connection - Generates the remarking surface by stitching together the surfaces of the leaf plate systems of the supported plates. It uses the existing geometry (as-is) of logical connections between the following objects:

- Supported plates and connected plates for plate remarking lines.
- Supported plates and connected profiles for profile remarking lines.
- Supported plates for seam remarking lines.

Based on Physical Connection - Generates the remarking surface by stitching the surfaces of the sides of the supported plate parts that face away from the jig floor. This option uses the physical connections between the following objects:

- Supported plates and connected plates for plate remarking lines.
- Supported plates and connected profiles for profile remarking lines.
- Supported plates for seam remarking lines.

The geometry for these remarking lines corresponds to the portion of the physical connection geometry that lies on the remarking surface.

Based on Part Surface and Logical Connection - Generates the remarking surface by stitching the surfaces of the sides of the supported plate parts that face away from the jig floor. This options uses the logical connections between the following objects:

- Supported plates and connected plates for plate remarking lines.
- Supported plates and connected profiles for profile remarking lines.
- Supported plates for seam remarking lines.

Seam Remarking Lines

Specifies whether seam remarking lines are applied or ignored. This value defaults to **Ignore** for individual plates. For assemblies, this value defaults to **Apply**.

Plate Remarking Lines

Specifies whether plate remarking lines are applied or ignored. The following options are available:

Ignore - Indicates that plate remarking lines are not applied.

Apply - Creates a remarking line at the logical connection.

Both Sides - Creates remarking lines representing the molded and anti-molded sides of the connected plate part.

Both Sides with LC - Creates the remarking lines representing the molded and anti-molded sides of the connected plate parts along with the logical connection remarking line.

Profile Remarking Lines

Specifies whether profile remarking lines are applied or ignored.

X-Axis Remarking Lines

Specifies whether frame remarking lines are applied or ignored.

Y-Axis Remarking Lines

Specifies whether longitudinal remarking lines are applied or ignored.

Z-Axis Remarking Lines

Specifies whether buttock remarking lines are applied or ignored.

Reference Curve Remarking Line

Specifies whether naval architecture reference curves are applied or ignored.

User Remarking Lines

Specifies whether user remarking lines are applied or ignored.

User Extensions

Specifies whether user extensions are applied or ignored.

Accuracy Check Points

Apply Rule

Specifies whether the display of accuracy check points is based on a rule, or is user defined. If this option is selected, all of the rules display as read-only. If this option is cleared, the rules are editable.

 **NOTE** This check box is only available during the creation of the pin jig.

Pin Jig Accuracy Check Rule Set

Specifies the rule set that the software uses to get the accuracy points. You can modify or add rule sets.

Remarking Bend Points

Specifies whether bend points are applied or ignored at remarking sharp bends.

Aft Boundary

Specifies the accuracy points on the Aft Boundary.

Lower Boundary

Specifies the accuracy points on the Lower Boundary.

Fore Boundary

Specifies the accuracy points on the Fore Boundary.

Upper Boundary

Specifies the accuracy points on the Upper Boundary.

Custom

Specifies the custom accuracy points.

Along Girth From Lower - Creates points starting from the lower boundary at a specified distance along the girth.

Along Girth From Upper - Creates points starting from the upper boundary at a specified distance along the girth.

Along Girth From Aft - Creates points starting from the aft boundary at a specified distance along the girth.

Along Girth From Fore - Creates points starting from the fore boundary at a specified distance along the girth.

At Pin Lines - Creates points at the intersection of pin rows and columns with the boundary.

At and Middle of Pin Lines - Creates points at the intersection of pin rows and columns with the boundary. Additional points are created at the middle of these points.

See Also

Pin Jig Properties Dialog Box (on page 194)

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type


Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

 **NOTE** You cannot define the filters using the **Configuration** tab.

Plant


Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.


Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

 **NOTE** The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

 **NOTE** You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified


Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

 **NOTE** Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

SECTION 8

Manufacturing Plate



Creates a two-dimensional wireframe representation of a detailed plate that has manufacturing data such as margins, shrinkage, marking lines, and beveling applied.

The plates that the software allows you to select are controlled by manufacturing plate rules that can be customized by your administrator.

NOTE You can specify the unfolding algorithm for the software to use by changing the **Surface Type Requested for Unfolding** property on the **General** tab of the **Manufacturing Plate Properties** dialog box. For more information, see *General Tab (Manufacturing Plate Properties Dialog Box)* (on page 206).

Manufacturing Plate Ribbon

Displays the controls used to create a manufacturing plate.



Properties

Activates the **Manufacturing Plate Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the manufacturing plate that you are about to place in the model. For more information, see *Manufacturing Plate Properties Dialog Box* (on page 206).



Plate Parts

Select the detailed plate part for which you want to create the manufacturing plate. The plate part must be a child of an assembly. You can select the part in a graphic view or in the **Workspace Explorer**.

Finish

Creates the manufacturing plate using the parameters that you have defined.



Cancel

Clears all selected plates.



Accept

Accepts all selected plates.

Process List

Select a plate process set from the list. A process is a rule-based geometrical definition of a manufacturing plate based on particular manufacturing information. For example, adding a margin to a plate edge is a process.





Marking List

Select a marking set from the list. A marking set is a group of rule-based marking line types that can be applied. For example, structural location marking lines, reference marking lines, and roll lines can be grouped together and applied.


What do you want to do?

- *Generate a manufacturing plate* (on page 205)
 - *Copy and paste a manufacturing plate* (on page 205)
 - *Modify a manufacturing plate* (on page 206)
 - *Delete a manufacturing plate* (on page 206)
-


Generate a manufacturing plate

1. Click **Manufacturing Plate** .
2. On the **Manufacturing Plate** ribbon, click **Plate Parts** .
3. Select the detailed plate parts for which you want to generate manufacturing plates.
TIP You can select the parts in the model, or from the **Workspace Explorer**.
4. Click **Accept** .
5. Select the process type from the **Process List**.
6. Select the marking set from the **Marking List**.
7. Click **Properties** , and change the default **Process** and **Marking** values if necessary.
8. Click **OK**.
9. Click **Finish**.



Copy and paste a manufacturing plate

1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing plate to copy.
TIPS
 - You can select the manufacturing plate in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Manufacturing Plates** to make the selection easier.
3. Click **Edit > Copy**.
4. Click **Edit > Paste**.
5. Select the plate on which to paste the manufacturing plate.
TIP You must select a non-manufactured, detailed plate for the destination.
6. Click **OK** on the **Paste** dialog box.

Modify a manufacturing plate

1. Click **Select**  on the vertical toolbar.
2. On the **Assembly** tab of the **Workspace Explorer**, select the manufacturing plate to modify.
NOTE Manufacturing plates do not have a graphic representation, so you cannot select them in the graphic view.
3. Using the ribbon controls, edit the manufacturing plate as needed.

Delete a manufacturing plate

1. Click **Select**  on the vertical toolbar.
2. On the **Assembly** tab of the **Workspace Explorer**, select the manufacturing plate to delete.
3. Click **Delete** .

Manufacturing Plate Properties Dialog Box

Specifies the properties for the manufacturing plate that you are editing.

See also

General Tab (Manufacturing Plate Properties Dialog Box) (on page 206)

Process Tab (Manufacturing Plate Properties Dialog Box) (on page 211)

Marking Tab (Manufacturing Plate Properties Dialog Box) (on page 214)

Cut/Mark Tab (Manufacturing Plate Properties Dialog Box) (on page 217)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

Status Tab (on page 217)

General Tab (Manufacturing Plate Properties Dialog Box)

Standard

Name

Specifies the name of the manufacturing plate.

Name Rule

Select the naming rule to use to name the manufacturing plate.

- Select **DefaultNameRule** to use the following syntax: <Parent Name (excluding block name)>-<Workshare Location ID>. For example, SH.12-1-1, where SH.12-1 is the parent name (the block name is taken out of the parent name), and 1 is the Workshare Location ID.
- Select **User Defined** to type a name in the **Name** box.

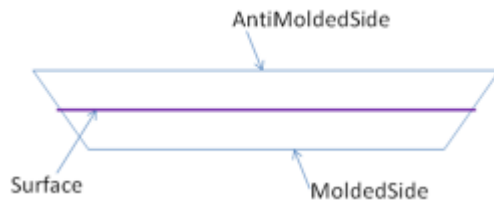
Manufacturing Plate Frame System

Specifies the frame system associated with the manufacturing plate.

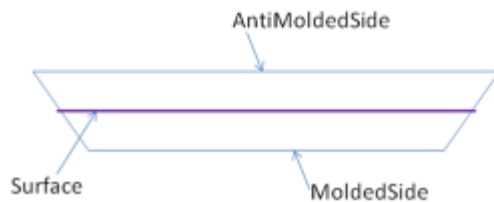
Surface Type Requested for Unfolding

Specifies the surface that the software uses when computing the unfolding. Available options include the following:

Detailing Neutral Surface - This is the default option. Detailing gets the unbounded reference surface without features, and then offsets to the required location based on the rule-given neutral axis offset value. The software intersects the extended surface with the plate solid to trim the surface within the boundaries. This surface considers all the aspects such as bounding elements and bevel parameters to give the exact part length.



Detailing Part Surface - This option is implemented in Structural Detailing. The selected part surface is retrieved before any features are applied, and then the surface is offset in the required direction. In this option, there is no trimming involved. Bevel offset parameters are applied, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.

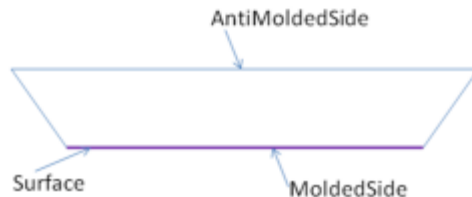


True Molded Surface - This option is implemented in Structural Manufacturing. It takes the immediate parent leaf plate system of the selected plate part, and gets its surface geometry. This geometry is then offset to the required surface. The part geometry is not used. The molded surface is offset according to the neutral axis ratio value. Bevel offset parameters are applied, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.


NOTE Shrinkage, margins, and bevels are applied as usual. The part lengths are not as accurate for bevels as is unfolding with the neutral surface, however, because the bevel correction is applied to the true molded surface contours. **Manufacturing Service Manager**, **Marking**, and **Marking by Definition** work as usual.


True Part Surface - This option is implemented in Structural Manufacturing. It takes the plate part surface before any features or chamfers, and then offsets that surface in the required direction. This option applies bevel offset parameters, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.

Detailing Molded Surface - This option is implemented in Structural Detailing. It takes the molded surface and intersects it with the part solid to give it the same shape as the part, including accounting for bounding part thicknesses. It should give the same result as **Detailing Neutral Surface**, except that it is computed in the molded surface instead of the neutral surface. This option fails in cases where it cannot successfully intersect the solid part with the molded system surface. Bevel offset parameters are applied, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.



For a plate bounded only by design or planning seams and other contours, there is no difference; the plate subsystem and the molded part surface are the same. However, if the part is bounded by any other plate systems or profiles, the surface accounts for the detailed shape and thickness of these boundaries. The plate subsystem does not. It is bounded only by the molded surfaces of its boundaries.

Imported Surface - This option is implemented in Structural Manufacturing. For this option, **Manufacturing Plate**  accepts a .sat file. This imported surface is used for generating contours and specifying the curves. All the marking lines are projected onto this imported surface, rather than the surface inherited from the part surface. Imported surface geometry is fed to the unfolding algorithm. Bevel corrections, margins, and shrinkages are applied on the resulting manufacturing data.

Import Bigger Reference Surface - This option is implemented in Structural Manufacturing. For this option, **Manufacturing Plate**  accepts a .sat file. The outer contours and markings are projected onto a larger reference surface. Imported surface geometry is fed to the unfolding algorithm. Bevel corrections, margins, and shrinkages are applied on the resulting manufacturing data. This option is similar to **Imported Surface** without using the boundary contours of the imported surface.

NOTE For non-planar surfaces, the software first tries to unfold the neutral surface by default. If there is a failure getting the neutral surface and its contour data, the software uses the molded side of the object and puts the object on the **To Do List** with a warning indicating that the unfolding used the molded side for processing. This process is followed for both planar and curved geometry.

Surface Type Used for Unfolding

Displays the surface type that was used to generate the unfolding data. This property is read-only.

Plate Tightness

Specifies the plate tightness associated with the manufacturing plate.

Naming Category

Displays the naming category associated with the manufacturing plate.

Length

Specifies the length of the manufacturing plate.

Width

Specifies the width of the manufacturing plate.

Curvature Type

Displays the curvature type of the manufacturing object. The ProfileCurvature codelist defines this list.

Plate Type

Specifies the plate type associated with the manufacturing plate.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools > Recompute Objects**. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Survivability Status

Specifies the status associated with the manufacturability object. The list is defined by the MfgState codelist.

Generated State - All the steps and rules were successfully applied, and the manufacturing part was generated based on the steps and rules.

Partial State - Not all steps or rules were applied successfully. The information is not complete and you must add information to pass the manufacturability check. After the check passes, the step moves to **Modified State**.

Modified State - You have provided additional information that could lead to acceptance of the manufacturability check. The part remains in a warning state on the **To Do List**. You can refresh the part so that it is regenerated. If the check is successful, it gets a new status.

Nesting**Export Date**

Displays the export date of the manufacturing object.

Registration Date

Displays the registration date of the manufacturing object.

Nesting Date

Displays the nesting date of the manufacturing object.

Lot Number

Displays the lot number of the manufacturing object.

Part Registered Number

Displays the part registered number of the manufacturing object.

Order Number

Displays the order number of the manufacturing object.

Lot Material Type

Displays the lot material type of the manufacturing object.

Lot Material Grade

Displays the lot material grade of the manufacturing object.

Lot Length

Displays the lot length of the manufacturing object.

Lot Width

Displays the lot width of the manufacturing object.

Lot Thickness

Displays the lot thickness of the manufacturing object.

Part Type

Specifies the part type of the manufacturing object.

Output Type

Displays the output type of the manufacturing object.

Output Status

Indicates whether or not the nesting system is up to date. This attribute is not currently used.

User Process

Displays user-defined pre-processes that the software applies to all of the selected plates.

See Also

Manufacturing Plate Properties Dialog Box (on page 206)

Process Tab (Manufacturing Plate Properties Dialog Box)

Processes are rule-based and can be customized by each manufacturing facility.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Results

Select a manufacturing plate process type from the list. This list is the same as the **Process List** on the manufacturing plate ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

Specifies modifiable parameter values.

Rule-Based

If the value of the parameter is rule-based, a check box displays. Clear the check box and then select a different parameter value if necessary.

Catalog Value

If you modified the value, the original catalog value for the process displays.

Parameters**Neutral Axis**

Some facilities make the neutral axis position a function of thickness. This determines the position based on material, thickness, and bending direction where there is no change in length when unwrapping.

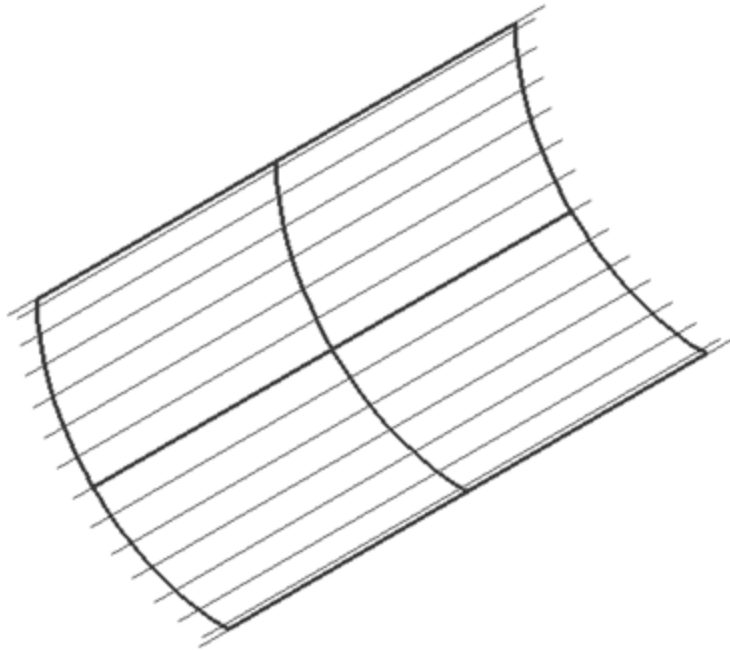
Upside

The side of the part that is facing up while it is being cut.

Unwrap Algorithm

Select the unwrap algorithm:

- **Developable** - Useful for unwrapping knuckled plates and single curved shell plates. Bending lines are evaluated on the curved surface.



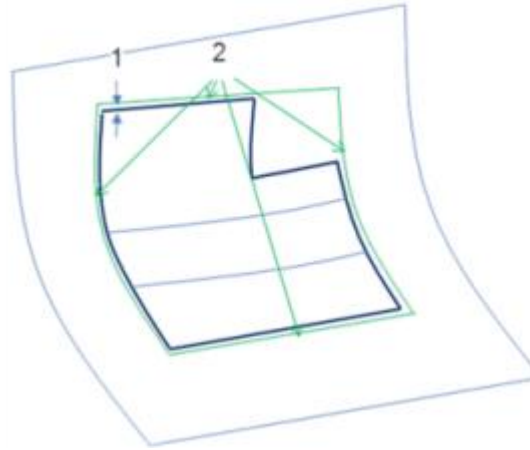
Unwrapping is done by *unbending* at the bending lines. The first two bending lines define the plane that the shell plate is unwrapped into. The unbending is then done by rotating a part of the shell plate around a bending line. By doing this rotation around all bending lines, the shell plate is unwrapped into a plane.

- **Flat** - Useful for unwrapping flat plates. There is no unfolding done with this algorithm. The contour curves and markings are simply transformed into a plane.
- **TwoDirUnfold** - Minimizes the change of curve length after unwrapping the curve to a plane. Because this approach is based on minimizing the change of curve length, it does not handle extremely curved surfaces (for example, bending angles larger than 120 degrees) which includes most closed surfaces. For a curved part with a great degree of curvature (such as near the nose), split the plate into several pieces before unwrapping it.
- **Geodesic_Merge** - Depends on n-surface merge as it uses iso-u and iso-v meshed points to evaluate normal and derivatives to derive local geodesic. The algorithm then uses the meshed points to derive one-to-many corresponding u-v parameters for unwrapping curves on the surface. The quality of this method largely depends on the accuracy of the merged surfaces and the parameterization. This method can handle closed surfaces if n-surface can generate a good result.

This method uses an approach similar to finite element modeling, where the surface to unfold is approximated into a regular mesh of points and normals. The objectives are to preserve the geodesic length and minimize the deformation of the mesh. This method produces accurate results when unfolding developable surfaces such as cylinders, cones, or developable ruled surfaces.

- **SNU** - Useful for unwrapping hull plates. All of the patches comprising the hull shape, along with the contour and marking data, is sent to a third-party unfolding algorithm developed by Seoul National University (SNU).

- **SNU_Single_Surface** - Similar to SNU, except that the patches are merged into a single b-spline surface before the software sends it to the SNU algorithm.
- **SNU_FourSided_10** - Adds material to all four sides to create a rectangle which provides a 10 mm margin around the original contours.



- 1 - Margin dimension
- 2 - Newly calculated contour

- **SNU_FourSided_50** - Adds material to all four sides to create a rectangle which provides a 50 mm margin around the original contours.
- **SNU_FourSided_100** - Adds material to all four sides to create a rectangle which provides a 100 mm margin around the original contours.

Bevel

The facilities need to handle the bevel information depending on their capabilities. They can have a continuous bevel, a bevel within increments, or a fixed bevel. The deviation angle determines when a segment of a physical connection should be created to provide new bevel information for that segment.

Roll Lines

Specifies the roll line calculation process.

User Defined Values

Shipyards can control certain values used during the unfold process of plate parts.

Weld Tab

Automatically specifies a weld tab based on the assigned custom rule. These can be partially solved by replacing the feature by an alternative representation defined for scallops.

Feature Tab

Automatically specifies a feature tab based on the process rule.

Knuckle Tab

Automatically specifies a knuckle tab based on the process rule.

Margin

Specifies margin values for the manufacturing object based on the process rule.

Shrinkage

Specifies shrinkage values for the manufacturing object based on the process rule.

Marking

Specifies marking lines for the manufacturing object based on the process rule.

Cut Mark Control

Controls the behavior of cut, mark, and ignore behavior for all features, openings, and hole traces.

Marking Validate

Controls the validation of marking lines. You can merge overlapping marking lines or connect disjointed markings.

Plate Custom

Stores custom manufacturing plate part attributes that will be exported to the final XML.


Feature

Shipyards control the way the plate features are processed during unwrap. Some openings must be unwrapped with the plate. The rest should be reapplied after the unwrap process is completed.

Some facilities are using this in their analysis software to determine if they are influencing the strength of a plate and therefore should or should not be included in the unwrap process.

Marking Tab (Manufacturing Plate Properties Dialog Box)

Manufacturing plate marking sets are rule-based and can be customized by facility.

 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a manufacturing plate marking set from the list. This list is the same as the **Marking Set** list on the manufacturing plate ribbon.

Parameter

Displays the parameters for the marking set that you have selected.

Value

You can modify the marking values if necessary.

Rule-Based

If the value of the marking set is rule-based, a check box displays. Clear the check box and then select a different marking value if necessary.

Catalog Value

If you modified the value, the original catalog value for the marking displays.

The following marking processes are considered to be included in the marking lists. This marking list is fixed and uses the codelist proposal of Yes/No values.

NOTE The rules for marking lines, and the symbols they place, are customizable. For example, you can customize the marking line rules to place marking lines at the left, right, or center of the connected part.

Marking Type	Comments
Plate Location	System generated parameter. This parameter marks the locations of the bounded plates.
Profile Location	System generated parameter. This parameter marks the location of stiffening profiles.
Collar Location	System generated parameter. This parameter marks the location of connected collar plates.
Bracket Location	System generated parameter. This parameter marks the location of connected bracket plates.
Lap Connection	System generated parameter. This parameter marks the location of lap connected plates.
End Connection	System generated parameter. This parameter marks the locations of end connected profiles.
Fitting Plate to Plate T Joint	Shipyards need to control the number of symbols, symbol type, and its parameters. Used for accuracy control when positioning connected parts.
Fitting Plate to Profile Penetration	Shipyards need to control the number of symbols, symbol type, and its parameters. Used for accuracy control when positioning connected parts.
Fitting Plate to Profile	Shipyards need to control the number of symbols, symbol type, and its parameters. Used for accuracy control when positioning connected parts.
Reference Planes Frame	System generated parameter. Displays the intersection of the plate and X-planes within the range of the plate.
Reference Planes WaterLine	System generated parameter. Displays the intersection of the plate and Y-planes within the range of the plate.
Reference Planes Buttock	System generated parameter. Displays the intersection of the plate and Z-planes within the range of the plate.
Block Control	User marking. Displays marks for accuracy control.
Seam Control	User marking. Displays marks for accuracy control.
Template Location Lines	System generated parameter. Marks the locations of templates.

Marking Type	Comments
Base Control Line	System generated parameter. Marks the location of template base control lines.
Roll Lines	System generated parameter. Marks the roll on a curved plate.
Knuckle Lines	System generated parameter. Marks the location of a knuckle.
Naval Architect Lines	System generated parameter. Marks the naval architect lines, such as flat-of-side or flat-of-bottom.
Roll Boundaries	System generated parameter. Marks the start and end of a roll on a curved plate.
Ship Direction	System generated parameter. Tick marks showing the global orientation of the plate.
Diagonals	System generated parameter. Displays the diagonals of the plate.
Robot Mark	System generated parameter. Displays robot marks generated with the User defined marking line option of the Marking command.
Painting Line	System generated parameter. Displays painting lines generated with the User defined marking line option of the Marking command.
Mark (ship name)	User marking. Marks the ship name of the plate.
Plate Edge Check Lines	System generated parameter. Displays marks near the plate edge for accuracy control.
Web Frames Check Lines	System generated parameter. Displays marks near the plate edge connected to the hull for accuracy control.
Bridge	This parameter is not currently used.
Feature Tab	This parameter is not currently used.
Welding Tab	This parameter is not currently used.
Bending Control Line	System generated parameter. Displays the bending control lines generated in the unfold process.
Margin	System generated parameter. Displays the pre-margin contour. For a negative margin, this lies outside the final plate contour and is not marked.
User Defined	User marking. Displays user-defined markings.

Cut/Mark Tab (Manufacturing Plate Properties Dialog Box)

Specifies the properties for adding, cutting, or marking conditions to the manufacturing plate that you are editing. By using the properties on this tab you can override the **PlateCutMarkControl** option available for manufacturing plates, profiles, and members.

Feature Name

Specifies the names of features, openings, and holes. When you select a feature on the dialog box, that feature highlights in the model.

Condition


Specifies the cut option for the feature when it is applied to the unfolded manufacturing part. The available options include **Cut**, **Mark**, or **Ignore**.

Set by Rule

Sets the cut condition for all selected features as specified by the rules. Select a feature row in the first column to use this button.

Status Tab

Specifies the status of the plate parts and physical connections contained in the manufacturing part or panel. When an object displays **Out of Date** for **Manufacturing Object Status** on the **General** tab, you can look at the **Status** tab to see which parts or connections are responsible.

You can perform updates with **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

Detailed Part

The plate part is detailed.

Light Part

The plate part is not detailed. No physical connections for the part are included in the manufacturing part definition. The manufacturing part is out of date.

Up to Date

The state of the physical connection is current in its relationships to the plate parts.

Deleted

The physical connection has been deleted because of a change in the detailed part or the deletion of the detailed part. The manufacturing part or panel is out of date.

Modified

The plate part has been modified and the manufacturing part or panel is out-of-date.



Added

The object was created after creation of the manufacturing part. The manufacturing part or panel is out of date.

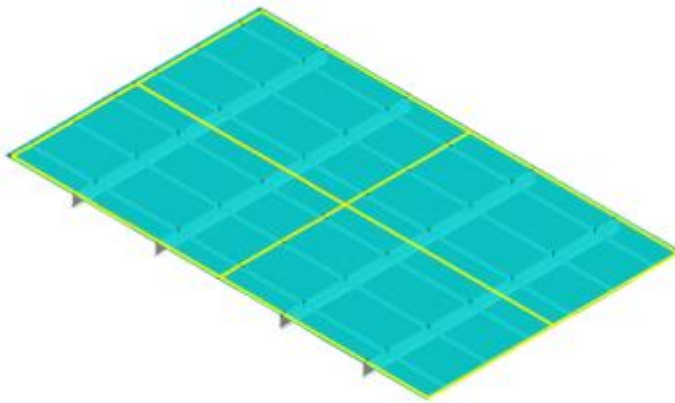
Removed

The object was removed after creation of the manufacturing part. The manufacturing part or panel is out of date.

Manufacturing Panel

 Creates a two-dimensional wireframe representation for detailed plate parts combined into a panel. Each representation contains manufacturing data such as margins, shrinkage, marking lines, and beveling to support panel line production. The output of a manufacturing panel can be saved to an .xml document accessible through **Managed Export** (on page 343) and **Manufacturing Service Manager (on page 317)** .

A panel is a collection of edge-to-edge plate parts that are welded together into one plate before other parts, such as plates, profiles, or brackets, are welded. All plate parts for the panel must be in the same planning assembly.



A manufacturing panel is similar to a manufacturing plate. A manufacturing plate contains a single plate, while a manufacturing panel contains a collection of welded plates. Manufacturing plates and manufacturing panels are used at different stages of the manufacturing process and have their own process and marking requirements.

NOTE You can specify the unfolding algorithm for the software to use by changing the **Surface Type Requested for Unfolding** property on the **General** tab of the **Manufacturing Panel Properties** dialog box. For more information, see *General Tab (Manufacturing Panel Properties Dialog Box)* (on page 223).

Manufacturing Panel Ribbon

Displays the controls used to create a manufacturing panel.

Properties

Opens the **Manufacturing Panel Properties** dialog box to view and modify the panel properties, which include process and marking properties. For more information, see *Manufacturing Panel Properties Dialog Box* (on page 223).

Select an Assembly or Block

Specifies the assembly containing the panel to manufacture. Select the assembly from the **Assembly** tab of the **Workspace Explorer**.

 **Select Plate Parts**

Specifies the plates to include in the manufacturing panel. Select the plates in a graphic view or in the **Workspace Explorer**.

Finish

Creates the manufacturing panel using the parameters that you have defined.

**Panel Plate List**

Opens the **Panel Plate List** dialog box to review and define the plates included in the panel. For more information, see *Panel Plate List Dialog Box* (on page 222). This option is not available when you are modifying an existing panel.

**Cancel**

Clears all selected plates.

**Accept**

Accepts all selected plates for inclusion in the manufacturing panel.

Process List

Selects a panel process set from the list. A process is a rule-based geometrical definition of a manufacturing panel based on particular manufacturing information. For example, adding a margin to a plate edge is a process.

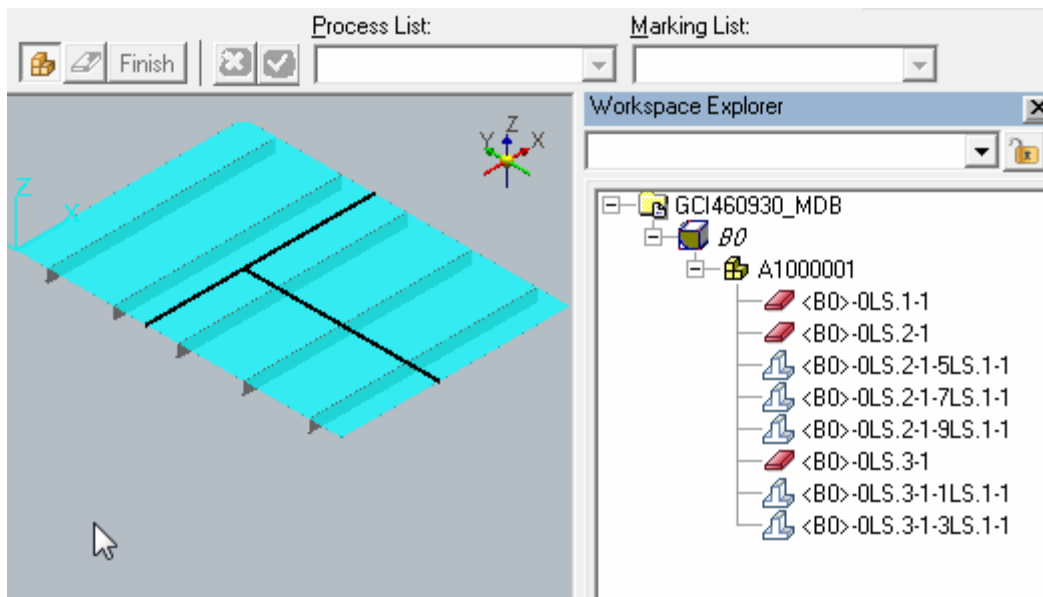
Marking List




Selects a marking set from the list. A marking set is a group of rule-based marking line types that can be applied. For example, structural location marking lines, reference marking lines, and roll lines can be grouped together and applied.

What do you want to do?




- *Generate a manufacturing panel* (on page 220)
 - *Display the plate parts in a manufacturing panel* (on page 221)
 - *Modify a manufacturing panel* (on page 222)
 - *Delete a manufacturing panel* (on page 222)
-

Generate a manufacturing panel

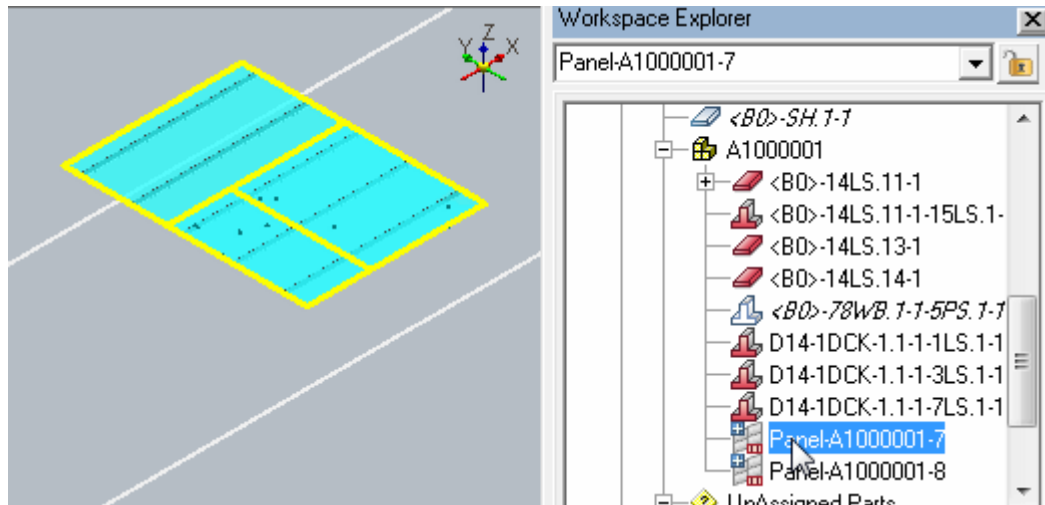


1. Click **Manufacturing Panel** .
The command ribbon displays.
2. Click **Select an Assembly or Block** , and then select an assembly  from the **Assembly** tab of the **Workspace Explorer**.
3. Select the detailed plate parts in the assembly to include in the panel.

TIPS

- You can select the parts in a graphic view or from the **Workspace Explorer**.
 - You can also click **Panel Plate List** , and then select the plates from the **Panel Plate List** dialog box.
4. Click **Accept** .
 5. Select the process type from the **Process List**.
 6. Select the marking set from the **Marking List**.
 7. If required, click **Properties** , change the default **Process** and **Marking** values, and then click **OK**.
 8. Click **Finish**.

The software creates the panel object under the assembly on the **Assembly** tab of the **Workspace Explorer**.



Display the plate parts in a manufacturing panel


1. In the Structural Manufacturing task, click **Select**  on the vertical toolbar.
2. On the **Assembly** tab of the **Workspace Explorer**, select the manufacturing panel to modify.

NOTE Manufacturing panels do not have a graphic representation, so you cannot select them in the graphic view.



The plate parts on the panel highlight in the graphic view and in the **Workspace Explorer**.




Modify a manufacturing panel

1. Click **Select**  on the vertical toolbar.
2. On the **Assembly** tab of the **Workspace Explorer**, select the manufacturing panel to modify.
NOTE Manufacturing panels do not have a graphic representation, so you cannot select them in the graphic view.
3. Using the ribbon controls, edit the manufacturing panel as required.

Delete a manufacturing panel

1. Click **Select**  on the vertical toolbar.
2. On the **Assembly** tab of the **Workspace Explorer**, select the manufacturing panel to delete.
3. Click **Delete** .

Panel Plate List Dialog Box

Displays the plates in the panel. You open this dialog box by clicking **Panel Plate List**  on the command ribbon.

Current List

Displays the currently selected plates for the panel.



Potential List

Expands the dialog box and opens the **Potential List**.



Add Parts to the Current List

Moves the selected plate in the **Potential List** to the **Current List**.



Remove Parts from the Current List

Removes the selected plate from the **Current List** and moves it to the **Potential List**.



Clear All Parts from the Current List

Removes all plates from the **Current List** and moves them to the **Potential List**.

Potential List

Displays other plates in the assembly. You can add plates to the panel by moving the plates from the **Potential List** to the **Current List**.

NOTE All plates must be edge-to-edge to form the panel.

Manufacturing Panel Properties Dialog Box

Specifies the properties for the manufacturing panel that you are editing.

See also

General Tab (Manufacturing Panel Properties Dialog Box) (on page 223)

Process Tab (Manufacturing Panel Properties Dialog Box) (on page 227)

Marking Tab (Manufacturing Panel Properties Dialog Box) (on page 230)

Cut/Mark Tab (Manufacturing Panel Properties Dialog Box) (on page 233)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

Status Tab (on page 217)

General Tab (Manufacturing Panel Properties Dialog Box)

Standard

Name

Specifies the name of the manufacturing panel.

Name Rule

Specifies the naming rule to use for the manufacturing panel.

- Select **DefaultNameRule** to use the following syntax: Panel-<Assembly Name>-<Sequence Number>. For example, Panel-A1000001-9, where A1000001 is the assembly name, and 9 indicates the ninth panel created in the model.
- Select **User Defined** to type a name in the **Name** box.

Frame System

Specifies the coordinate system associated with the manufacturing plate. Select **Inherit from Assembly Parent**, or select an existing model coordinate system, such as **CS_0**. **Inherit from Assembly Parent** is the default value.

Common Cut Master

Specifies the plate part in the manufacturing panel used to trigger the process and marking rules. The software uses the plate part geometry with the rules defined in the **Process List** and **Marking List** boxes of the **Manufacturing Panel** ribbon.

Surface Type Requested for Unfolding

Specifies the surface that the software uses when computing the unfolding. Available options include the following:

Detailing Neutral Surface - This is the default option. Detailing gets the unbounded reference surface without features, and then offsets to the required location based on the rule-given neutral axis offset value. The software intersects the extended surface with the plate solid to trim the surface within the boundaries. This surface considers all the aspects such as bounding elements and bevel parameters to give the exact part length.



Detailing Part Surface - This option is implemented in Structural Detailing. The selected part surface is retrieved before any features are applied, and then the surface is offset in the required direction. In this option, there is no trimming involved. Bevel offset parameters are applied, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.

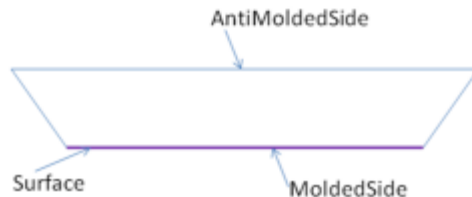


True Molded Surface - This option is implemented in Structural Manufacturing. It takes the immediate parent leaf plate system of the selected plate part, and gets its surface geometry. This geometry is then offset to the required surface. The part geometry is not used. The molded surface is offset according to the neutral axis ratio value. Bevel offset parameters are applied, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.


NOTE Shrinkage, margins, and bevels are applied as usual. The part lengths are not as accurate for bevels as is unfolding with the neutral surface, however, because the bevel correction is applied to the true molded surface contours. **Manufacturing Service Manager**, **Marking**, and **Marking by Definition** work as usual.


True Part Surface - This option is implemented in Structural Manufacturing. It takes the plate part surface before any features or chamfers, and then offsets that surface in the required direction. This option applies bevel offset parameters, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.

Detailing Molded Surface - This option is implemented in Structural Detailing. It takes the molded surface and intersects it with the part solid to give it the same shape as the part, including accounting for bounding part thicknesses. It should give the same result as **Detailing Neutral Surface**, except that it is computed in the molded surface instead of the neutral surface. This option fails in cases where it cannot successfully intersect the solid part with the molded system surface. Bevel offset parameters are applied, but there is a considerable difference in part lengths because it does not trim the offset surface to lateral faces.



For a plate bounded only by design or planning seams and other contours, there is no difference; the plate subsystem and the molded part surface are the same. However, if the part is bounded by any other plate systems or profiles, the surface accounts for the detailed shape and thickness of these boundaries. The plate subsystem does not. It is bounded only by the molded surfaces of its boundaries.

Imported Surface - This option is implemented in Structural Manufacturing. For this option, **Manufacturing Plate**  accepts a .sat file. This imported surface is used for generating contours and specifying the curves. All the marking lines are projected onto this imported surface, rather than the surface inherited from the part surface. Imported surface geometry is fed to the unfolding algorithm. Bevel corrections, margins, and shrinkages are applied on the resulting manufacturing data.

Import Bigger Reference Surface - This option is implemented in Structural Manufacturing. For this option, **Manufacturing Plate**  accepts a .sat file. The outer contours and markings are projected onto a larger reference surface. Imported surface geometry is fed to the unfolding algorithm. Bevel corrections, margins, and shrinkages are applied on the resulting manufacturing data. This option is similar to **Imported Surface** without using the boundary contours of the imported surface.

NOTE For non-planar surfaces, the software first tries to unfold the neutral surface by default. If there is a failure getting the neutral surface and its contour data, the software uses the molded side of the object and puts the object on the **To Do List** with a warning indicating that the unfolding used the molded side for processing. This process is followed for both planar and curved geometry.

Surface Type Used for Unfolding

Displays the surface type that was used to generate the unfolding data. This property is read-only.

Length

Displays the length of the manufacturing panel. This property is read-only.

Width

Displays the width of the manufacturing panel. This property is read-only.

Curvature Type

Displays the curvature type of the manufacturing object. The ProfileCurvature codelist defines

this list.

Plate Type

Displays the plate type associated with the plate parts in the manufacturing panel. Plate type is inherited from the parent plate system.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools > Recompute Objects**. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using **Manufacturing Service Manager** . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Survivability Status

Specifies the status associated with the manufacturability object. The list is defined by the MfgState codelist.

Generated State - All the steps and rules were successfully applied, and the manufacturing part was generated based on the steps and rules.

Partial State - Not all steps or rules were applied successfully. The information is not complete and you must add information to pass the manufacturability check. After the check passes, the step moves to **Modified State**.

Modified State - You have provided additional information that could lead to acceptance of the manufacturability check. The part remains in a warning state on the **To Do List**. You can refresh the part so that it is regenerated. If the check is successful, it gets a new status.

Process Tab (Manufacturing Panel Properties Dialog Box)

Results

Specifies a manufacturing panel process type from the list. This list is the same as the **Process List** on the manufacturing panel ribbon. Clear the rule-based box to select a different process type.

Parameter

Displays the parameters for the selected process.

Value

Specifies modifiable parameter values.


Rule-Based

Displays a check box if the value of the parameter is rule-based. Clear the check box and then select a different parameter value if required.

Catalog Value

Displays the original catalog value for the process if you modified the value.

Parameters

 **NOTE** A manufacturing panel uses the same process parameters as a manufacturing plate.

Neutral Axis

Some facilities make the neutral axis position a function of thickness. This determines the position based on material, thickness, and bending direction where there is no change in length when unwrapping.

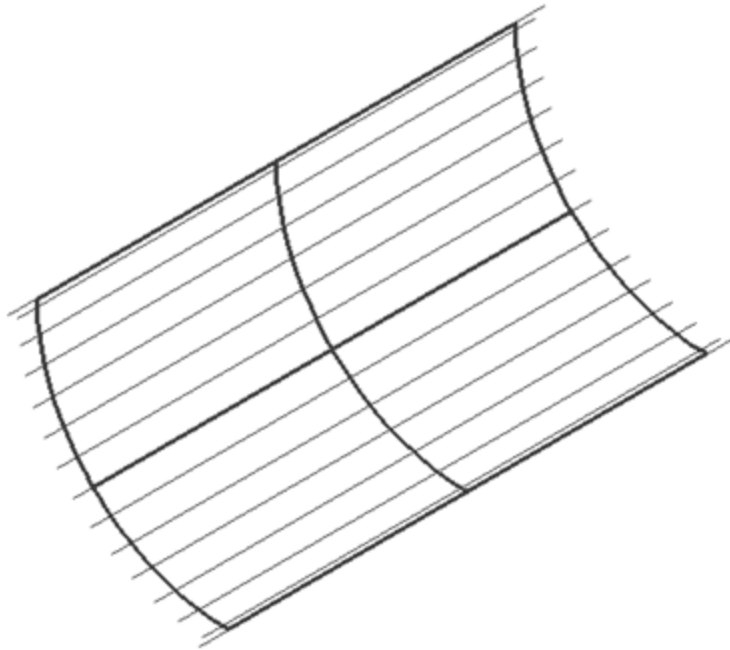
Upside

The side of the part that is facing up while it is being cut.

Unwrap Algorithm

Select the unwrap algorithm:

- **Developable** - Useful for unwrapping knuckled plates and single curved shell plates. Bending lines are evaluated on the curved surface.



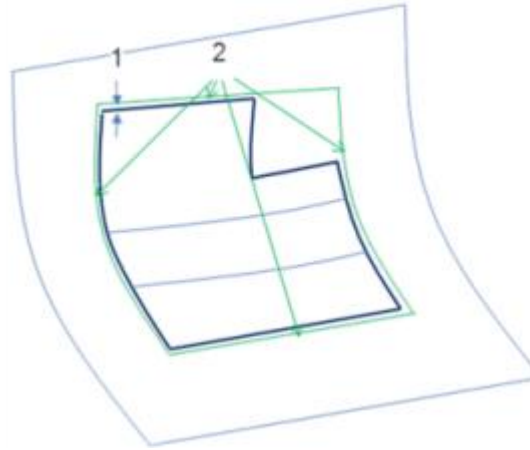
Unwrapping is done by *unbending* at the bending lines. The first two bending lines define the plane that the shell plate is unwrapped into. The unbending is then done by rotating a part of the shell plate around a bending line. By doing this rotation around all bending lines, the shell plate is unwrapped into a plane.

- **Flat** - Useful for unwrapping flat plates. There is no unfolding done with this algorithm. The contour curves and markings are simply transformed into a plane.
- **TwoDirUnfold** - Minimizes the change of curve length after unwrapping the curve to a plane. Because this approach is based on minimizing the change of curve length, it does not handle extremely curved surfaces (for example, bending angles larger than 120 degrees) which includes most closed surfaces. For a curved part with a great degree of curvature (such as near the nose), split the plate into several pieces before unwrapping it.
- **Geodesic_Merge** - Depends on n-surface merge as it uses iso-u and iso-v meshed points to evaluate normal and derivatives to derive local geodesic. The algorithm then uses the meshed points to derive one-to-many corresponding u-v parameters for unwrapping curves on the surface. The quality of this method largely depends on the accuracy of the merged surfaces and the parameterization. This method can handle closed surfaces if n-surface can generate a good result.

This method uses an approach similar to finite element modeling, where the surface to unfold is approximated into a regular mesh of points and normals. The objectives are to preserve the geodesic length and minimize the deformation of the mesh. This method produces accurate results when unfolding developable surfaces such as cylinders, cones, or developable ruled surfaces.

- **SNU** - Useful for unwrapping hull plates. All of the patches comprising the hull shape, along with the contour and marking data, is sent to a third-party unfolding algorithm developed by Seoul National University (SNU).

- **SNU_Single_Surface** - Similar to SNU, except that the patches are merged into a single b-spline surface before the software sends it to the SNU algorithm.
- **SNU_FourSided_10** - Adds material to all four sides to create a rectangle which provides a 10 mm margin around the original contours.



- 1 - Margin dimension
- 2 - Newly calculated contour

- **SNU_FourSided_50** - Adds material to all four sides to create a rectangle which provides a 50 mm margin around the original contours.
- **SNU_FourSided_100** - Adds material to all four sides to create a rectangle which provides a 100 mm margin around the original contours.

Bevel

The facilities need to handle the bevel information depending on their capabilities. They can have a continuous bevel, a bevel within increments, or a fixed bevel. The deviation angle determines when a segment of a physical connection should be created to provide new bevel information for that segment.

Roll Lines

Specifies the roll line calculation process.

User Defined Values

Shipyards can control certain values used during the unfold process of plate parts.

Weld Tab

Automatically specifies a weld tab based on the assigned custom rule. These can be partially solved by replacing the feature by an alternative representation defined for scallops.

Feature Tab

Automatically specifies a feature tab based on the process rule.

Knuckle Tab

Automatically specifies a knuckle tab based on the process rule.

Margin

Specifies margin values for the manufacturing object based on the process rule.

Shrinkage

Specifies shrinkage values for the manufacturing object based on the process rule.

Marking

Specifies marking lines for the manufacturing object based on the process rule.

Cut Mark Control

Controls the behavior of cut, mark, and ignore behavior for all features, openings, and hole traces.

Marking Validate

Controls the validation of marking lines. You can merge overlapping marking lines or connect disjointed markings.

Plate Custom

Stores custom manufacturing plate part attributes that will be exported to the final XML.

Feature

Shipyards control the way the plate features are processed during unwrap. Some openings must be unwrapped with the plate. The rest should be reapplied after the unwrap process is completed.

Some facilities are using this in their analysis software to determine if they are influencing the strength of a plate and therefore should or should not be included in the unwrap process.

Marking Tab (Manufacturing Panel Properties Dialog Box)

Results

Specifies a manufacturing panel marking set from the list. This list is the same as the **Marking List** on the manufacturing panel ribbon. Clear the rule-based check box to select a different marking type.

Parameter

Displays the parameters for the selected marking set.

Value

Specifies modifiable marking values.

Rule-Based

Displays a check box if the value of the marking set is rule-based. Clear the check box and then select a different marking value if necessary.


Catalog Value

Displays the original catalog value for the marking if you modified the value.

Parameters

A manufacturing panel uses the same marking parameters as a manufacturing plate.

Manufacturing plate marking sets are rule-based and can be customized by facility.

 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected,

that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a manufacturing plate marking set from the list. This list is the same as the **Marking Set** list on the manufacturing plate ribbon.

Parameter

Displays the parameters for the marking set that you have selected.

Value

You can modify the marking values if necessary.

Rule-Based

If the value of the marking set is rule-based, a check box displays. Clear the check box and then select a different marking value if necessary.

Catalog Value

If you modified the value, the original catalog value for the marking displays.

The following marking processes are considered to be included in the marking lists. This marking list is fixed and uses the codelist proposal of Yes/No values.

NOTE The rules for marking lines, and the symbols they place, are customizable. For example, you can customize the marking line rules to place marking lines at the left, right, or center of the connected part.

Marking Type	Comments
Plate Location	System generated parameter. This parameter marks the locations of the bounded plates.
Profile Location	System generated parameter. This parameter marks the location of stiffening profiles.
Collar Location	System generated parameter. This parameter marks the location of connected collar plates.
Bracket Location	System generated parameter. This parameter marks the location of connected bracket plates.
Lap Connection	System generated parameter. This parameter marks the location of lap connected plates.
End Connection	System generated parameter. This parameter marks the locations of end connected profiles.
Fitting Plate to Plate T Joint	Shipyards need to control the number of symbols, symbol type, and its parameters. Used for accuracy control when positioning connected parts.

Marking Type	Comments
Fitting Plate to Profile Penetration	Shipyards need to control the number of symbols, symbol type, and its parameters. Used for accuracy control when positioning connected parts.
Fitting Plate to Profile	Shipyards need to control the number of symbols, symbol type, and its parameters. Used for accuracy control when positioning connected parts.
Reference Planes Frame	System generated parameter. Displays the intersection of the plate and X-planes within the range of the plate.
Reference Planes WaterLine	System generated parameter. Displays the intersection of the plate and Y-planes within the range of the plate.
Reference Planes Buttock	System generated parameter. Displays the intersection of the plate and Z-planes within the range of the plate.
Block Control	User marking. Displays marks for accuracy control.
Seam Control	User marking. Displays marks for accuracy control.
Template Location Lines	System generated parameter. Marks the locations of templates.
Base Control Line	System generated parameter. Marks the location of template base control lines.
Roll Lines	System generated parameter. Marks the roll on a curved plate.
Knuckle Lines	System generated parameter. Marks the location of a knuckle.
Naval Architect Lines	System generated parameter. Marks the naval architect lines, such as flat-of-side or flat-of-bottom.
Roll Boundaries	System generated parameter. Marks the start and end of a roll on a curved plate.
Ship Direction	System generated parameter. Tick marks showing the global orientation of the plate.
Diagonals	System generated parameter. Displays the diagonals of the plate.
Robot Mark	System generated parameter. Displays robot marks generated with the User defined marking line option of the Marking command.
Painting Line	System generated parameter. Displays painting lines generated with the User defined marking line option of the Marking command.
Mark (ship name)	User marking. Marks the ship name of the plate.

Marking Type	Comments
Plate Edge Check Lines	System generated parameter. Displays marks near the plate edge for accuracy control.
Web Frames Check Lines	System generated parameter. Displays marks near the plate edge connected to the hull for accuracy control.
Bridge	This parameter is not currently used.
Feature Tab	This parameter is not currently used.
Welding Tab	This parameter is not currently used.
Bending Control Line	System generated parameter. Displays the bending control lines generated in the unfold process.
Margin	System generated parameter. Displays the pre-margin contour. For a negative margin, this lies outside the final plate contour and is not marked.
User Defined	User marking. Displays user-defined markings.

Cut/Mark Tab (Manufacturing Panel Properties Dialog Box)

Specifies the properties for adding, cutting, or marking conditions to the manufacturing panel. Use the properties on this tab to override the **PlateCutMarkControl** option.

Feature Name

Specifies the names of features, openings, and holes. When you select a feature on the dialog box, that feature highlights in the model.

Condition

Specifies the cut option for the feature when it is applied to the unfolded manufacturing panel. The available options include **Cut**, **Mark**, or **Ignore**.

Set by Rule

Sets the cut condition for all selected features as specified by the rules. Select a feature row in the first column to use this option.

SECTION 9

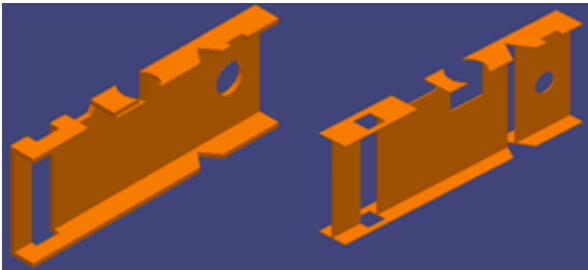
Manufacturing Profile



Creates a two-dimensional wireframe representation of a detailed profile that has manufacturing data applied such as margins, shrinkage, marking lines, and beveling.

The profiles that the software allows you to select are controlled by manufacturing profile rules that can be customized by your administrator.

Structural Manufacturing supports features on manufacturing profiles that split either the web or flange into disjointed pieces such as those shown in the following figure.



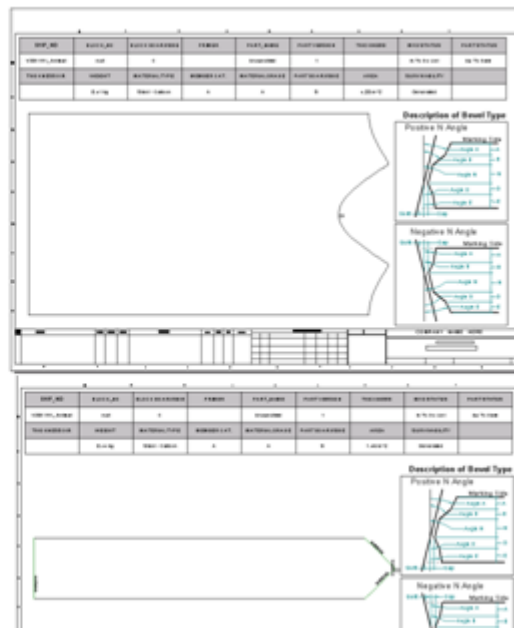
You can unfold multi-face sections such as **SquareBar**, **RoundBar**, and **HalfRoundBar**. Unfolding of closed cross section types can be controlled by the **ProfileUnfold** option. If the **ProfileUnfold** option is set to **Apply**, round bar is unfolded like a wrap-around paper template. If **ProfileUnfold** is set to **Ignore**, a projected view of the cross section is generated as manufacturing output.

ProfileUnfold
– Apply



– Ignore

 **NOTES**



- For built-up profiles, you can generate the profile XML for the web and flange independently of one another.
- The software automatically generates fitting marks on the web and flange of built-up profiles.

Manufacturing Profile Ribbon

Displays the controls used to create a manufacturing profile.

Properties

Activates the **Manufacturing Profile Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the manufacturing profile that you are about to place in the model. For more information, see *Manufacturing Profile Properties Dialog Box* (on page 238).

Profile Part

Select the detailed profile part for which you want to create the manufacturing profile. You can select the part in a graphic view or in the **Workspace Explorer**.

Finish

Creates the manufacturing profile using the parameters that you have defined.

Cancel

Clears all selected profiles.

Accept

Accepts all selected profiles.


Process List

Select a template process from the list. A process is a rule-based geometrical definition of a profile based on particular manufacturing information. For example, adding a margin is a process.

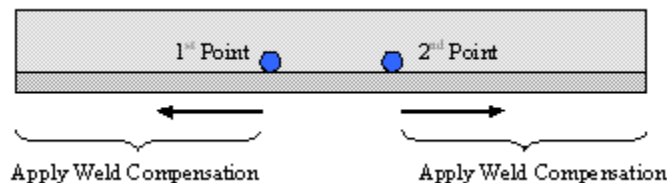
Marking List

Select a marking set from the list. A marking set is a group of rule-based marking line types that can be applied to a manufacturing profile. For example, structural location marking lines, reference marking lines, and roll lines can be grouped together and applied.

Weld Compensation Modify Ribbon

Displays controls to edit weld compensation settings. This ribbon appears only after you select an existing weld compensation object  on the **Assembly** tab in the **Workspace Explorer**.

You can define the weld compensation starting points and specify weld compensation values for the ends to which weld compensation is applied.



 **Properties**

Activates the **Weld Compensation Settings** dialog box, which you use to view and modify the settings for the weld compensation object you have selected. For more information, see *Weld Compensation Settings Dialog Box* (on page 247).

 **Part**

This command is disabled.

 **1st Point**

Allows you to set the first weld compensation starting point. The software highlights the part where you can set the point. The default location is the midpoint.

 **2nd Point**

Allows you to set the second weld compensation starting point. The software highlights the part where you can set the point.

Finish

Modifies the weld compensation object using the parameters that you have defined.

 **Cancel**

Clears the weld compensation starting point.





 **Accept**


Accepts the weld compensation starting point.

What do you want to do?


- *Generate a manufacturing profile* (on page 236)
 - *Copy and paste a manufacturing profile* (on page 237)
 - *Modify a manufacturing profile* (on page 237)
 - *Delete a manufacturing profile* (on page 237)
 - *Enable and set weld compensation* (on page 238)
-

Generate a manufacturing profile


1. Click **Manufacturing Profile** .
2. On the **Manufacturing Profile** ribbon, click **Profile Parts** .
3. Select the detailed profile parts for which you want to generate manufacturing profiles.
 **TIP** You can select the parts in the model, or from the **Workspace Explorer**.
4. Click **Accept** .
5. Select the process type from the **Process List**.

6. Select a marking set from the **Marking Set**.
7. Click **Properties**  and change the default **Process** and **Marking** values if necessary.
8. Click **Finish**.



Copy and paste a manufacturing profile

1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing profile to copy.
TIPS
 - You can select the manufacturing profile in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Manufacturing Profile Part** to make the selection easier.
3. Click **Edit > Copy**.
4. Click **Edit > Paste**.
5. Select the profile on which to paste the manufacturing profile.
TIP You must select a non-manufactured, detailed profile for the destination.
6. Click **OK** on the **Paste** dialog box.

Modify a manufacturing profile


1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing profile to modify.
TIP You can select the profile in the model, or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the manufacturing profile as needed.


Delete a manufacturing profile






1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing profile to delete.
3. Click **Delete** .

Enable and set weld compensation

1. Enable weld compensation for the part.
 - a. Go to the **Process Tab (Manufacturing Profile Properties Dialog Box)** (on page 242) of the manufacturing profile properties or the **Process** tab of the manufacturing built-up member properties.
 - b. Set the **Weld Compensation** process to **Apply** or **User-defined**.
 - c. Click **Finish**.

A weld compensation object  is created for the part.

2. Select the weld compensation object  on the **Assembly** tab in the **Workspace Explorer**.
*The **Weld Compensation Modify** ribbon is displayed.*

3. Click **Properties**  to view and edit weld compensation settings.
4. Set the first weld compensation starting point by clicking **1st Point**  and clicking on a point on the highlighted part. The default location is the midpoint.
5. Click **Accept**  to confirm the first point.
6. Set the second weld compensation starting point by clicking **2nd Point**  and clicking on a point on the highlighted part.
7. Click **Accept**  to confirm the second point.
8. Click **Finish**.

The weld compensation is applied from the starting points towards the end of the part.

Manufacturing Profile Properties Dialog Box

Specifies the properties for the manufacturing profile that you are editing.

See Also

General Tab (Manufacturing Profile Properties Dialog Box) (on page 239)

Process Tab (Manufacturing Profile Properties Dialog Box) (on page 242)

Marking Tab (Manufacturing Profile Properties Dialog Box) (on page 244)

Cut/Mark Tab (Manufacturing Plate Properties Dialog Box) (on page 217)

Nesting Tab (Manufacturing Profile Properties Dialog Box) (on page 245)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Manufacturing Profile Properties Dialog Box)

Standard

Name

Specifies the name of the manufacturing profile.

Name Rule

Select the naming rule to use to name the manufacturing profile.

- Select **DefaultNameRule** to use the following syntax: <Parent Name (excluding block name)>-<Workshare Location ID>. For example, SH.11-1-PS.1-1-1, where SH.11-1-PS.1-1 is the parent name (the block name is taken out of the parent name), and 1 is the Workshare Location ID.
- Select **User Defined** to type a name in the **Name** box.

Frame System

Specifies the frame system associated with the manufacturing profile. This value indicates whether the coordinate system is inherited from the parent object, or is set to a coordinate system that has frames in range of the object.

Survivability Status

Displays the survivability status of the manufacturing object.

- **Generated** - The system generated the manufacturing object.
- **Edited** - The user has edited the manufacturing object's geometry.
- **Annotated** - The user has edited the manufacturing object's annotations.
- **PartInformation Edited** - The manufacturing object's part information has been edited.

Error Sequence State

Specifies the reason for a manufacturing command failure.

Curvature Type

Displays the curvature type of the manufacturing object. The ProfileCurvature codelist defines this list.

Profile Type

Displays the profile type of the manufacturing profile.

Is Twisted

Indicates whether the object is twisted or not twisted. The list is defined by the StrMfgIsTwisted codelist.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools** >

Recompute Objects. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using

Manufacturing Service Manager . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Length

Displays the length of the manufacturing profile.

Width (Web)

Displays the width of the web of the manufacturing profile.

Width (Flange)

Displays the width of the flange of the manufacturing profile.

Thickness (Web)

Displays the thickness of the web of the manufacturing profile.

Thickness (Flange)

Displays the thickness of the flange of the manufacturing profile.

Dimensions

Length Structural Detail Web

Specifies an approximate length of the detailed web for the detailed manufacturing object.

Length Structural Detail Top Flange

Specifies an approximate length of the top flange for the detailed manufacturing object.

Length Structural Detail Bottom Flange

Specifies an approximate length of the bottom flange for the detailed manufacturing object.

Length Structural Detail Total

Specifies the total length of the detailed manufacturing object.

NOTE While there are four different lengths that display on the property page, only **Length Structural Detail Total** displays the detailing length.

Length Unfolded Web

Specifies the web length of the unfolded manufacturing object.

Length Unfolded Top Flange

Specifies the top flange length of the unfolded manufacturing object.

Length Unfolded Bottom Flange

Specifies the bottom flange length of the unfolded manufacturing object.

Length Unfolded Total

Specifies the total length of the unfolded manufacturing object.

NOTE The unfolded length values are the lengths of the extended objects after they are unfolded but before any features are applied. This length has almost no meaning to the designer because the extended length could be arbitrarily long. If the unfolded length is shorter than **Length Structural Detail Total**, the unfolding process may have been unsuccessful. Because the object is extended as a whole, the summation of the unfolded lengths for the individual faces equal the total unfolded length. Because most stiffeners do not have a bottom flange, the bottom flange unfolded length is typically zero. The same holds for stiffeners that do not have a top flange.

Length Before Features Web

Specifies the web length before features for the manufacturing object.

Length Before Features Top Flange

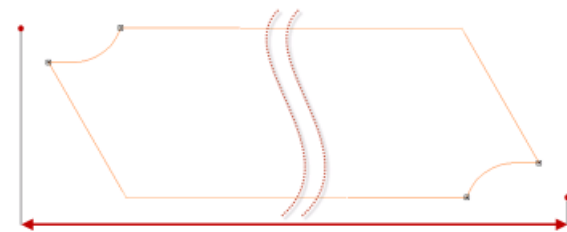
Specifies the top flange length before features for the manufacturing object.

Length Before Features Bottom Flange

Specifies the bottom flange length before features for the manufacturing object.

Length Before Features Total

Specifies the total length before features for the manufacturing object.



NOTE The **Before Features** values display the length that must be cut off the object stock

before it can be suitably cut at the ends. This is the length of the object before the end cuts remove material from the profile solid.

The lengths for each of the faces, as well as the total length, are all determined by the end-cuts themselves which internally define *length points*. This is illustrated with the red dot in the previous figure. The two length points that are farthest apart from each face define the **Before Features** length for that face. Likewise, the two length points of the whole object that are farthest from each other go to define the **Before Features** length of the whole object.

Length After Features Web

Specifies the web length after features for the manufacturing object.

Length After Features Top Flange

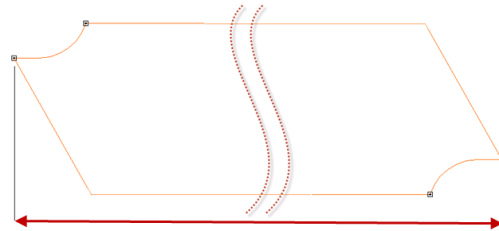
Specifies the top flange length after features for the manufacturing object.

Length After Features Bottom Flange

Specifies the bottom flange length after features for the manufacturing object.

Length After Features Total

Specifies the total length after features for the manufacturing object.



NOTE The **After Features** lengths display the distance between the points of the object that represent the extremity of material. This is computed after the end-cuts remove the material from the stiffener solid.

User Process

Displays user-defined pre-processes that the software applies to all of the selected plates.

See Also

Manufacturing Profile Properties Dialog Box (on page 238)

Process Tab (Manufacturing Profile Properties Dialog Box)

Processes are rule-based and can be customized by each facility.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a manufacturing plate process type from the list. This list is the same as the **Process List** on the manufacturing profile ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

You can modify the parameter values if necessary.

Rule-Based

If the value of the parameter is rule-based, a check box displays. Clear the check box and then select a different parameter value if necessary.

Catalog Value

If you modified the value, the original catalog value for the process displays.

Process Name	Comments
Upside	The side of the part that is facing <i>up</i> while it is being cut.
Profile Neutral Axis	The neutral axis is the dimension at which the profile length does not change regardless of its curvature.
Inverse Bending Line	Shipyards need to control the inverse bending line process. This process is executed if the profile is within the specified curvature limit.
Weld Tab	This is processed automatically based on custom rules. These rules can be partially solved by replacing the feature by an alternative representation defined for scallops.
Coaming Unwrap	It is required to control coamings unwrap. This is only applicable for profile edge reinforcements.
Clamp	This is required to allow you to set a certain clamp value on a given face.
Bevel	Allows you to select between different methods of setting the bevel deviation angle. This angle determines how a varying bevel is split.
User Defined Values	Shipyards can control certain values used during the unfold process of profile parts.
Unfold	Allows you to not unbend built-up profiles. The web can be cut to shape instead of bent.
Manufacture as Plate	If this process is set to apply, then separate plate XML data is generated for the web and each flange. If it is set to ignore, then one set of profile XML data is generated for the entire profile.
Weld Compensation	Allows you to select between different methods of computing weld compensation to account for the deformation on manufactured parts (such as buildup profiles, web frame plate, and plates with edge reinforcement), due to welding operations performed on them. If set, a weld compensation object is created for the part with a generalized set of properties that represent weld compensation information. Select the weld compensation object to modify the properties. For more information, see <i>Enable and set weld compensation</i> (on page 238).

See Also

Manufacturing Profile Properties Dialog Box (on page 238)

Marking Tab (Manufacturing Profile Properties Dialog Box)

Profile marking sets are rule-based and can be customized by each facility.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a manufacturing profile marking set from the list. This list is the same as the **Marking Set** list on the manufacturing profile ribbon.

Parameter

Displays the parameters for the marking set that you have selected.

Value

You can modify the marking values if necessary.

Rule-Based

If the value of the marking set is rule-based, a check box displays. Clear the check box and then select a different marking value if necessary.

Catalog Value

If you modified the value, the original catalog value for the marking displays.

NOTE The rules for marking lines, and the symbols they place, are customizable. For example, you can customize the marking line rules to place marking lines at the left, right, or center of the connected part.

Marking Type	Comments
User Mark	This is a user-defined mark.
Location	Location line of the profile.
Lap Connection	When profiles overlap, the lap area needs to be marked.
End Connection	When other profiles are bounded to the profile, the end connection needs to be marked.
Fitting Mark Profile	Fitting marks of the profile relative to the location on the plate.
Block Control	This is a user-defined mark.
Reference Planes Frame	Reference plane intersection marks in X direction.

Reference Planes WaterLine	Reference plane intersection marks in Z direction.
Reference Planes Buttock	Reference plane intersection marks in Y direction.
Seam Control	Seam control mark, when the profile is split.
Roll Boundaries	Roll boundary mark in case of a curved profile.
Ship Direction	The profile orientation relative to the ship.
Margin	Marking of the margin if applied.
Inverse Bending Line	Inverse bending lines which become straight after bending.
Knuckle Lines	Knuckle location mark.
Profile Hole	This marking type is not currently used.
Built-up PC	Generates a line on the flange or web showing the location of the connected web or flange of a built-up profile.

See Also

Manufacturing Profile Properties Dialog Box (on page 238)

Nesting Tab (Manufacturing Profile Properties Dialog Box)

Displays the nesting properties of the object. This tab is only available when you create a MfgNestData object by sending the part to nesting and receiving feedback from the nesting system.

Export Date

Displays the export date of the manufacturing object.

Registration Date

Displays the registration date of the manufacturing object.

Nesting Date

Displays the nesting date of the manufacturing object.

Lot Number

Displays the lot number of the manufacturing object.

Part Registered Number

Displays the part registered number of the manufacturing object.

Order Number

Displays the order number of the manufacturing object.

Lot Material Type

Displays the lot material type of the manufacturing object.

Lot Material Grade

Displays the lot material grade of the manufacturing object.

Lot Length

Displays the lot length of the manufacturing object.

Lot Width

Displays the lot width of the manufacturing object.

Lot Thickness

Displays the lot thickness of the manufacturing object.

Part Type

Specifies the part type of the manufacturing object.

Output Type

Displays the output type of the manufacturing object.

Output Status

Indicates whether or not the nesting system is up to date. This attribute is not currently used.

Assembly Path

Displays the assembly path of the located object. The assembly path automatically updates when the part is exported and moved to a new location.

Part GUID

Displays the part lookup service for the nesting system. Creating the nested data object sets this value, and it should not be changed.

Nesting System

Displays the nesting system.

Routing

Displays production routing information.

Deletion Timestamp

Displays when the object is deleted.

Reference 1–Custom Attribute

Displays a user-defined value.

Reference 2–Custom Attribute

Displays a user-defined value.

Reference 3–Custom Attribute

Displays a user-defined value.

Number of Confirmed Parts

Displays the number of confirmed common parts in the nesting system.

Number of Nested Parts

Displays the number of common parts that are nested on a lot.

Weld Compensation Settings Dialog Box

Specifies the properties for the weld compensation object that you are editing.

See Also

Weld Compensation Tab (Weld Compensation Settings Dialog Box) (on page 247)

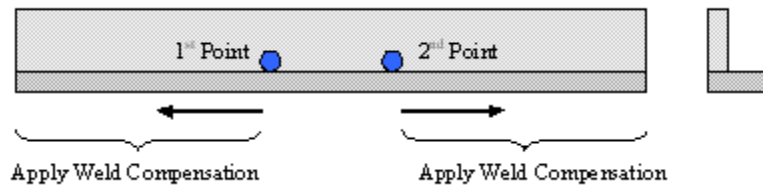
Configuration Tab (on page 86)

Notes Tab (on page 88)

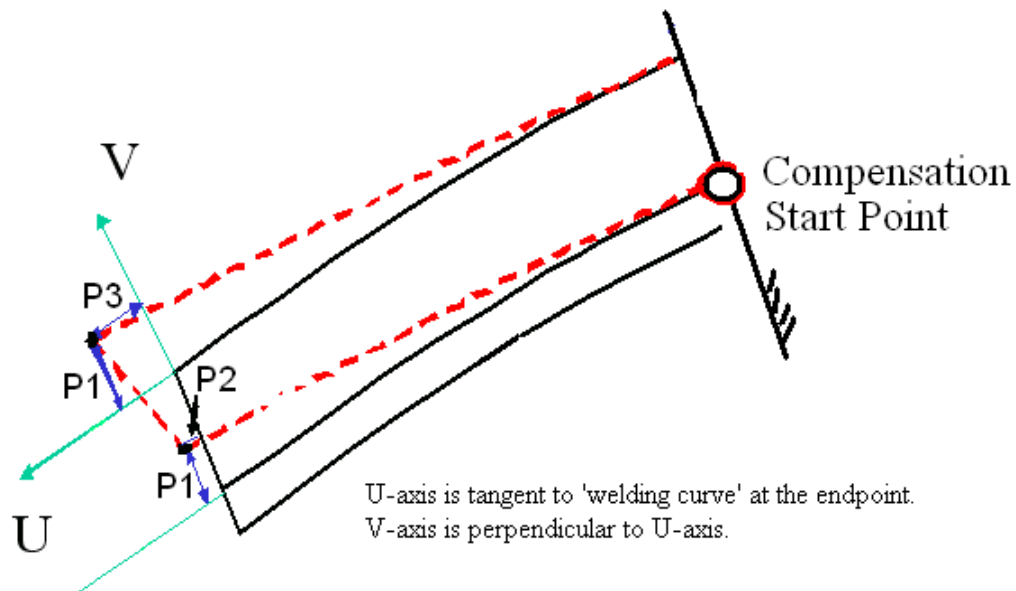
Weld Compensation Tab (Weld Compensation Settings Dialog Box)

Category

Select the end to which weld compensation is applied. You can select **WeldCompensation1**, which represents the end from the 1st point towards the start of the weld, or **WeldCompensation2**, which represents the end from the 2nd point towards the end of the weld.

**P1, P2, P3**

The weld compensation parameters as shown below.




Value

Specify a weld compensation value for **P1**, **P2**, and **P3** parameters.

Configuration Tab

Displays the creation, modification, and status information about an object.

 **NOTE** You cannot define the filters using the **Configuration** tab.

Plant


Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.


Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

 **NOTE** The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

 **NOTE** You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified


Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

 **NOTE** Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note


Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

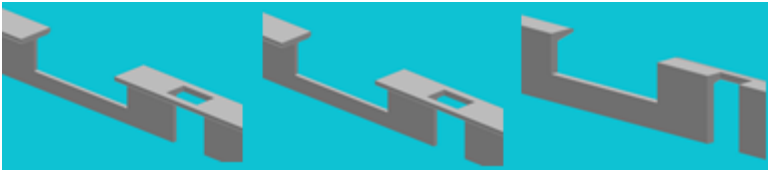
SECTION 10

Manufacturing Member

 Creates a two-dimensional wireframe representation of a detailed member that has manufacturing data such as margins, shrinkage, marking lines, and beveling applied.

The members that the software allows you to select are controlled by the CMfgProfile_MemberFilter rule. This rule also controls the members that are selected for manufacturing when you select an assembly. This rule is delivered in the *[Product Folder] StructManufacturing\Data\Rules\MfgPlate\Filter\MfgMemberCustomFilter.vbp*. The project is compiled into MfgMemberCustomFilter-Ref.dll and delivered in the **Symbols** share in the *[Reference Data Product Folder]\bin\StructManufacturing* folder. The rules are also bulkloaded into the Catalog.

Structural Manufacturing supports features on manufacturing members profiles that split either the web or flange into disjointed pieces such as those shown in the following figure.



NOTES

- For built-up members, you can generate the profile XML for the web and flange independently of one another.
- The software automatically generates fitting marks on the web and flange of built-up members.

Manufacturing Member Ribbon

Displays the controls used to create a manufacturing member.

Properties

Activates the **Manufacturing Member Properties** dialog box, which you use to view and modify the properties, including process and marking properties, of the manufacturing member that you are about to place in the model. For more information, see *Manufacturing Member Properties Dialog Box* (on page 255).

Member Part

Select the detailed member part for which you want to create the manufacturing member. You can select the part in a graphic view or in the **Workspace Explorer**.

Finish

Creates the manufacturing member using the parameters that you have defined.

Cancel

Clears the current selection.

✓ Accept

Accepts the current selection.


Process List

Select a template process from the list. A process is a rule-based geometrical definition of a member based on particular manufacturing information. For example, adding a margin is a process.

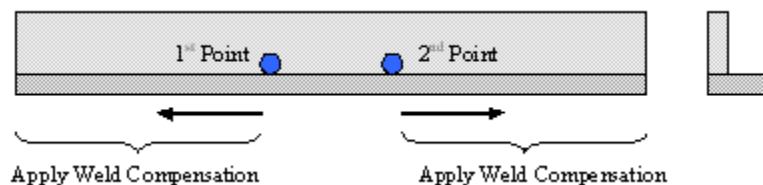
Marking List

Select a marking set from the list. A marking set is a group of rule-based marking line types that can be applied to a manufacturing member. For example, structural location marking lines, reference marking lines, and roll lines can be grouped together and applied.

Weld Compensation Modify Ribbon

Displays controls to edit weld compensation settings. This ribbon appears only after you select an existing weld compensation object  on the **Assembly** tab in the **Workspace Explorer**.

You can define the weld compensation starting points and specify weld compensation values for the ends to which weld compensation is applied.

** Properties**

Activates the **Weld Compensation Settings** dialog box, which you use to view and modify the settings for the weld compensation object you have selected. For more information, see *Weld Compensation Settings Dialog Box* (on page 247).

 Part

This command is disabled.

 1st Point

Allows you to set the first weld compensation starting point. The software highlights the part where you can set the point. The default location is the midpoint.

 2nd Point

Allows you to set the second weld compensation starting point. The software highlights the part where you can set the point.

Finish

Modifies the weld compensation object using the parameters that you have defined.

 Cancel

Clears the weld compensation starting point.





✔ Accept

Accepts the weld compensation starting point.


What do you want to do?

- *Generate a manufacturing member* (on page 253)
 - *Copy and paste a manufacturing member* (on page 253)
 - *Modify a manufacturing member* (on page 254)
 - *Delete a manufacturing member* (on page 254)
 - *Enable and set weld compensation* (on page 238)
-

Generate a manufacturing member


1. Click **Manufacturing Member** .
2. On the **Manufacturing Member** ribbon, click **Select Member Parts** .
3. Select the detailed member parts for which you want to generate manufacturing members.
TIP You can select the parts in the model, or from the **Workspace Explorer**.
4. Click **Accept** .
5. Select the process type from the **Process List**.
6. Select a marking set from the **Marking Set**.
7. Click **Properties**  and change the default **Process** and **Marking** values if necessary.
8. Click **Finish**.

Copy and paste a manufacturing member



1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing member to copy.
TIPS
 - You can select the manufacturing member in the model, or from the **Workspace Explorer**.
 - You can set the **Locate Filter** to **Manufacturing Member Part** to make the selection easier.
3. Click **Edit > Copy**.
4. Click **Edit > Paste**.
5. Select the detailed member part on which to paste the manufacturing member.
TIP You must select a non-manufactured, member for the destination.

6. Click **OK** on the **Paste** dialog box.

Modify a manufacturing member








1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing member to modify.
TIP You can select the member in the model, or from the **Workspace Explorer**.
3. Using the ribbon controls, edit the manufacturing member as needed.

Delete a manufacturing member

1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing member to delete.
3. Click **Delete** .

Enable and set weld compensation

1. Enable weld compensation for the part.
 - a. Go to the **Process Tab (Manufacturing Profile Properties Dialog Box)** (on page 242) of the manufacturing profile properties or the **Process** tab of the manufacturing built-up member properties.
 - b. Set the **Weld Compensation** process to **Apply** or **User-defined**.
 - c. Click **Finish**.

A weld compensation object  is created for the part.
2. Select the weld compensation object  on the **Assembly** tab in the **Workspace Explorer**.
*The **Weld Compensation Modify** ribbon is displayed.*
3. Click **Properties**  to view and edit weld compensation settings.
4. Set the first weld compensation starting point by clicking **1st Point**  and clicking on a point on the highlighted part. The default location is the midpoint.
5. Click **Accept**  to confirm the first point.
6. Set the second weld compensation starting point by clicking **2nd Point**  and clicking on a point on the highlighted part.
7. Click **Accept**  to confirm the second point.
8. Click **Finish**.

The weld compensation is applied from the starting points towards the end of the part.

Manufacturing Member Properties Dialog Box

Specifies the properties for the manufacturing member that you are editing.

See Also

General Tab (Manufacturing Member Properties Dialog Box) (on page 255)

Process Tab (Manufacturing Member Properties Dialog Box) (on page 259)

Marking Tab (Manufacturing Member Properties Dialog Box) (on page 260)

Cut/Mark Tab (Manufacturing Plate Properties Dialog Box) (on page 217)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Manufacturing Member Properties Dialog Box)

Standard

Name

Specifies the name of the manufacturing member.

Name Rule

Select the naming rule to use to name the manufacturing member. Select **User Defined** to type a name in the **Name** box.

Frame System

Specifies the frame system associated with the manufacturing member. This value indicates whether the coordinate system is inherited from the parent object or is set to a coordinate system that has frames in range of the object.

Length

Displays the length of the manufacturing member.

Width (Web)

Displays the width of the web of the manufacturing member.

Width (Flange)

Displays the width of the flange of the manufacturing member.

Thickness (Web)

Displays the thickness of the web of the manufacturing member.

Thickness (Flange)

Displays the thickness of the flange of the manufacturing member.

Curvature Type

Displays the curvature type of the manufacturing object. The ProfileCurvature codelist defines this list.

Is Twisted

Indicates whether the object is twisted or not twisted. The list is defined by the StrMfgIsTwisted codelist.

Detailed Part Status

Displays the state of the parent part or parts for the manufacturing object. This property only displays when you are modifying a part.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as connected parts. A relationship change can cause this state. You can update the detailed part by using **Tools > Recompute Objects**. For more information, see *Recompute Objects* in the *Common User's Guide*.

In Error - The part encountered an error. A relationship change such as connected parts can cause the error. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Warning - The part encountered a warning. A relationship change such as connected parts can cause the warning. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.


Light Part - The part has not been detailed.

Manufacturing Object Status

Displays the status of the manufacturing object. This property only displays when you are modifying a manufacturing object.

Up to Date - The part state is current with all of its relationships, such as connected parts.

Out of Date - The part state is not current with all of its relationships, such as detailed parts. A relationship change can cause this state. You can update the manufacturing part by using

Manufacturing Service Manager . For more information, see *Manufacturing Service Manager* (on page 317).

In To Do List - The part encountered an error or warning. A relationship change such as connected parts can cause this state. You can clear the error by using **View > To Do List**. For more information, see *To Do List* in the *Common User's Guide*.

Up to Date (Pre-nesting) - The state of a pre-nested part is current with all of its relationships, such as connected parts.

Out of Date (Pre-nesting) - The state of a pre-nested part is not current with all of its relationships, such as connected parts. A relationship change can cause this state.

In To Do List (Pre-nesting) - The pre-nested part encountered an error. A relationship change can cause this state.

 **NOTE** For more information on the pre-nesting service, see *Pre-Nesting* (on page 367).

Dimensions

Structural Detail Web

Specifies an approximate length of the detailed web for the detailed manufacturing object.

Structural Detail Top Flange

Specifies an approximate length of the top flange for the detailed manufacturing object.

Structural Detail Bottom Flange

Specifies an approximate length of the bottom flange for the detailed manufacturing object.

Structural Detail Total

Specifies the total length of the detailed manufacturing object.

NOTE While there are four different lengths that display on the property page, only **Length Structural Detail Total** displays the detailing length.

Unfolded Web

Specifies the web length of the unfolded manufacturing object.

Unfolded Top Flange

Specifies the top flange length of the unfolded manufacturing object.

Unfolded Bottom Flange

Specifies the bottom flange length of the unfolded manufacturing object.

Unfolded Total

Specifies the total length of the unfolded manufacturing object.

NOTE The unfolded length values are the lengths of the extended objects after they are unfolded but before any features are applied. This length has almost no meaning to the designer because the extended length could be arbitrarily long. If the unfolded length is shorter than **Length Structural Detail Total**, the unfolding process may have been unsuccessful. Because the object is extended as a whole, the summation of the unfolded lengths for the individual faces equal the total unfolded length. Because most stiffeners do not have a bottom flange, the bottom flange unfolded length is typically zero. The same holds for stiffeners that do not have a top flange.

Before Features Web

Specifies the web length before features for the manufacturing object.

Before Features Top Flange

Specifies the top flange length before features for the manufacturing object.

Before Features Bottom Flange

Specifies the bottom flange length before features for the manufacturing object.

Before Features Total

Specifies the total length before features for the manufacturing object.



NOTE The **Before Features** values display the length that must be cut off the object stock before it can be suitably cut at the ends. This is the length of the object before the end cuts remove material from the profile solid.

The lengths for each of the faces, as well as the total length, are all determined by the end-cuts themselves which internally define *length points*. This is illustrated with the red dot in the previous figure. The two length points that are farthest apart from each face define the

Before Features length for that face. Likewise, the two length points of the whole object that are farthest from each other go to define the **Before Features** length of the whole object.

After Features Web

Specifies the web length after features for the manufacturing object.

After Features Top Flange

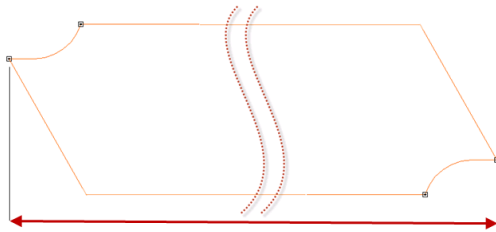
Specifies the top flange length after features for the manufacturing object.

After Features Bottom Flange

Specifies the bottom flange length after features for the manufacturing object.

Length After Features Total

Specifies the total length after features for the manufacturing object.



NOTE The **After Features** lengths display the distance between the points of the object that represent the extremity of material. This is computed after the end-cuts remove the material from the stiffener solid.

Nesting

Export Date

Displays the export date of the manufacturing object.

Registration Date

Displays the registration date of the manufacturing object.

Nesting Date

Displays the nesting date of the manufacturing object.

Lot Number

Displays the lot number of the manufacturing object.

Part Registered Number

Displays the part registered number of the manufacturing object.

Order Number

Displays the order number of the manufacturing object.

Lot Material Type

Displays the lot material type of the manufacturing object.

Lot Material Grade

Displays the lot material grade of the manufacturing object.

Lot Length

Displays the lot length of the manufacturing object.

Lot Width

Displays the lot width of the manufacturing object.

Lot Thickness

Displays the lot thickness of the manufacturing object.

Part Type

Specifies the part type of the manufacturing object.

Output Type

Displays the output type of the manufacturing object.

Output Status

Indicates whether or not the nesting system is up to date. This attribute is not currently used.

See Also

Manufacturing Member Properties Dialog Box (on page 255)

Process Tab (Manufacturing Member Properties Dialog Box)

Processes are rule-based and can be customized by each facility.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a manufacturing member process type from the list. This list is the same as the **Process List** on the manufacturing member ribbon.

Parameter

Displays the parameters for the process that you have selected.

Value

You can modify the parameter values if necessary.

Rule-Based

If the value of the parameter is rule-based, a check box displays. Clear the check box and then select a different parameter value if necessary.

Catalog Value

If you modified the value, the original catalog value for the process displays.

Process Name	Comments
Upside	The side of the part that is facing <i>up</i> while it is being cut.
Neutral Axis	The neutral axis is the dimension at which the member length does not

	change regardless of its curvature.
Inverse Bending Line	Shipyards need to control the inverse bending line process. This process is executed if the member is within the specified curvature limit.
Weld Tab	This is processed automatically based on custom rules. These rules can be partially solved by replacing the feature by an alternative representation defined for scallops.
Coaming Unwrap	It is required to control coamings unwrap. This is only applicable for member edge reinforcements.
Clamp	This is required to allow you to set a certain clamp value on a given face.
Bevel	Allows you to select between different methods of setting the bevel deviation angle. This angle determines how a varying bevel is split.
User Defined Values	Shipyards can control certain values used during the unfold process of member parts.
Unfold	Allows you to not unbend built-up profiles. The web can be cut to shape instead of bent.
Manufacture as Plate	If this process is set to apply, then separate plate XML data is generated for the web and each flange. If it is set to ignore, then one set of profile XML data is generated for the entire profile.
Weld Compensation	Allows you to select between different methods of computing weld compensation to account for the deformation on manufactured parts (such as buildup profiles, web frame plate, and plates with edge reinforcement), due to welding operations performed on them. If set, a weld compensation object is created for the part with a generalized set of properties that represent weld compensation information. Select the weld compensation object to modify the properties. For more information, see <i>Enable and set weld compensation</i> (on page 238).

See Also

Manufacturing Member Properties Dialog Box (on page 255)

Marking Tab (Manufacturing Member Properties Dialog Box)

Member marking sets are rule-based and can be customized by each facility.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

User Answers

Select a manufacturing member marking set from the list. This list is the same as the **Marking Set** list on the manufacturing member ribbon.

Parameter

Displays the parameters for the marking set that you have selected.

Value


You can modify the marking values if necessary.

Rule-Based

If the value of the marking set is rule-based, a check box displays. Clear the check box and then select a different marking value if necessary.

Catalog Value

If you modified the value, the original catalog value for the marking displays.

 **NOTE** The rules for marking lines, and the symbols they place, are customizable. For example, you can customize the marking line rules to place marking lines at the left, right, or center of the connected part.

Marking Type	Comments
User Mark	This is a user-defined mark.
Location	Location line of the profile.
Lap Connection	When profiles overlap, the lap area needs to be marked.
End Connection	When other profiles are bounded to the profile, the end connection needs to be marked.
Fitting Mark Profile	Fitting marks of the profile relative to the location on the plate.
Block Control	This is a user-defined mark.
Reference Planes Frame	Reference plane intersection marks in X direction.
Reference Planes WaterLine	Reference plane intersection marks in Z direction.
Reference Planes Buttock	Reference plane intersection marks in Y direction.
Seam Control	Seam control mark, when the profile is split.
Roll Boundaries	Roll boundary mark in case of a curved profile.
Ship Direction	The profile orientation relative to the ship.
Margin	Marking of the margin if applied.
Inverse Bending Line	Inverse bending lines which become straight after bending.

Knuckle Lines	Knuckle location mark.
Profile Hole	This marking type is not currently used.
Built-up PC	Generates a line on the flange or web showing the location of the connected web or flange of a built-up profile.

See Also

Manufacturing Member Properties Dialog Box (on page 255)

Weld Compensation Settings Dialog Box

Specifies the properties for the weld compensation object that you are editing.

See Also

Weld Compensation Tab (Weld Compensation Settings Dialog Box) (on page 247)

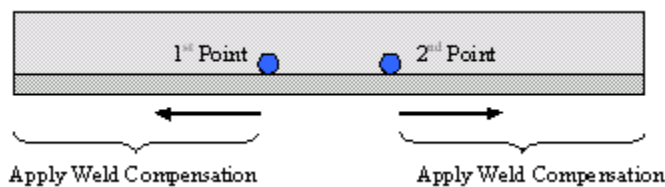
Configuration Tab (on page 86)

Notes Tab (on page 88)

Weld Compensation Tab (Weld Compensation Settings Dialog Box)

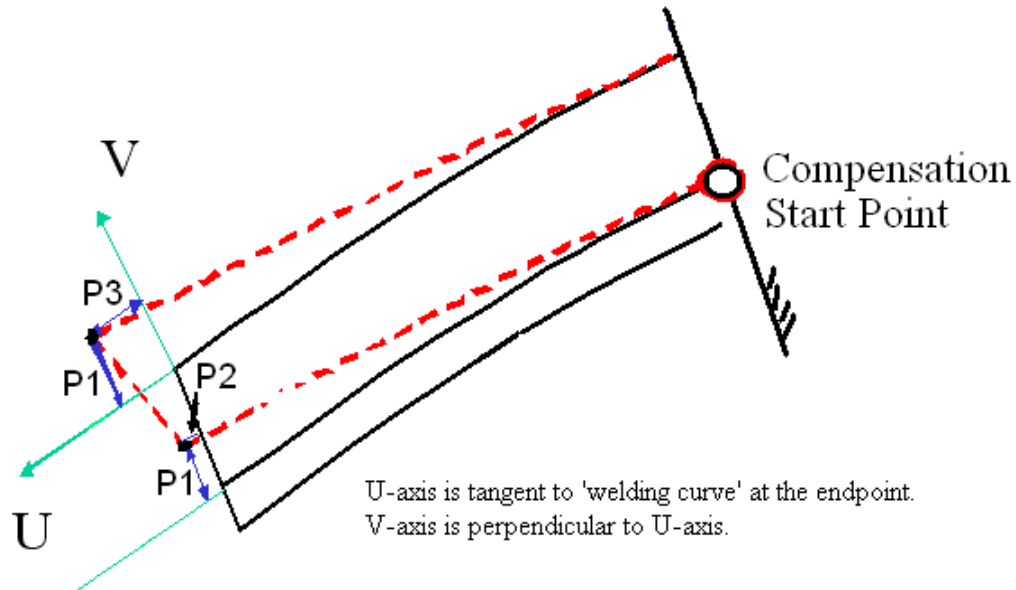
Category

Select the end to which weld compensation is applied. You can select **WeldCompensation1**, which represents the end from the 1st point towards the start of the weld, or **WeldCompensation2**, which represents the end from the 2nd point towards the end of the weld.



P1, P2, P3

The weld compensation parameters as shown below.



Value

Specify a weld compensation value for **P1**, **P2**, and **P3** parameters.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified


Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

 **NOTE** Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

SECTION 11

Batch Service Manager

Schedules a service manager batch job. This command is available on the right-click menu in the **Workspace Explorer** for **Blocks** and **Assemblies**. Each job has an associated XML file which contains the model database, the job name, the database provider, and the server name. Service Manager batch jobs are run on the local computer.

You can run the **Batch Service Manager** command either from within the **Smart 3D** environment or as a separate executable. If you run it as a separate executable, it always performs an update on the entire model.



This command creates a batch log file for each job. The filename of the log file is stored in the database, and the files are written to the %TEMP% folder. The information for the job is written to the STRMFGBatchProcessInfo table in the Model database.

This command uses the **Schedule Wizard** to specify the parameters for the batch job.

What do you want to do?



- *Schedule a manufacturing service manager batch job* (on page 266)
 - *Edit a manufacturing service manager batch job* (on page 267)
 - *Delete a manufacturing service manager batch job* (on page 267)
-

Schedule a manufacturing service manager batch job




1. Right-click a block or assembly in the **Workspace Explorer**.
2. Click **Batch Service Manager** on the right-click menu.
3. Click **Filter objects**, and specify the object types to include in the batch job.
 **TIP** For more information, see the *Select Filter Dialog Box* (on page 268).
4. Click **Schedule the batch job**.
 **TIP** For more information on the first page of the **Schedule Wizard**, see *Schedule Wizard - Page 1* (on page 268).
5. Click **Next**.
6. Click **Schedule a new job**.
7. Select the batch job for which you want to create a schedule from the list, and then click **Next**.
8. Specify the frequency at which you want to run the batch job, and then click **Next**.
9. Specify the time at which you want to start the batch job, and then click **Next**.
10. Review the batch job information. If you need to make changes, click **Back** to display the page you need to change. When the information meets your needs, click **Finish**.

11. Type the user name and password for the batch job on the **Enter User Name and Password** dialog box.

Edit a manufacturing service manager batch job

1. Right-click a block or assembly in the **Workspace Explorer**.
2. Click **Batch Service Manager** on the right-click menu.
3. Click **Filter objects**, and specify the object types to include in the batch job.
 **TIP** For more information, see the *Select Filter Dialog Box* (on page 268).
4. Click **Schedule the batch job**.
 **TIP** For more information on the first page of the **Schedule Wizard**, see *Schedule Wizard - Page 1* (on page 268).
5. Click **Next**.
6. Click **Edit an existing job**.
7. Select the batch job that you want to edit from the list, and then click **Next**.
8. Specify the frequency at which you want to run the batch job, and then click **Next**.
9. Specify the time at which you want to start the batch job, and then click **Next**.
10. Review the batch job information. If you need to make changes, click **Back** to display the page you need to change. When the information meets your needs, click **Finish**.
11. Type the user name and password for the batch job on the **Enter User Name and Password** dialog box.

Delete a manufacturing service manager batch job

1. Right-click a block or assembly in the **Workspace Explorer**.
2. Click **Batch Service Manager** on the right-click menu.
3. Click **Filter objects**, and specify the object types to include in the batch job.
 **TIP** For more information, see *Select Filter Dialog Box* (on page 268).
4. Click **Schedule the batch job**.
 **TIP** For more information on the first page of the **Schedule Wizard**, see *Schedule Wizard - Page 1* (on page 268).
5. Click **Next**.
6. Select the batch job that you want to delete from the list, and then click  **Delete**.
7. Review the batch job information. If you need to make changes, click **Back** to display the page you need to change. When the information meets your needs, click **Finish**.

See Also

Schedule Wizard - Page 1 (on page 268)
Schedule Wizard - Page 2 (on page 270)
Schedule Wizard - Page 3 (on page 270)

Schedule Wizard - Page 4 (on page 270)

Select Filter Dialog Box (on page 268)

Enter User Name and Password Dialog Box (on page 269)

Schedule Wizard - Page 1

Displays the controls used to schedule a batch job to update and review manufacturing objects.

Ship Name

Displays the name of the model. The software reads this information from the workspace if you are running the wizard from within Smart 3D.

Action

Determines whether the Manufacturing Service Manager updates the objects or updates the status.

Permission Group

Displays the permission group associated with the batch job. The software reads this information from the workspace if you are running the wizard from within Smart 3D.

Job Name

Displays the job name associated with the batch job. The software reads this information from the workspace if you are running the wizard from within Smart 3D.

Submit the batch job now

Starts the batch job when you finish the wizard.

Schedule the batch job

Starts the batch job at a time that you specify.

Filter objects

Displays the **Select Filter** dialog box. For more information, see *Select Filter Dialog Box* (on page 268).

Back

Displays the previous page in the wizard.

Next

Displays the next page in the wizard.

Finish

Displays the **Enter user name and password** dialog box. For more information, see *Enter User Name and Password Dialog Box* (on page 269). After you supply the user name and password, the Wizard closes and the batch job runs as scheduled.

See Also

Schedule Wizard - Page 2 (on page 270)

Select Filter Dialog Box

Displays the controls used to specify the object types to include in a Manufacturing Service Manager batch job.

MfgPlate

Includes manufacturing plates.

MfgProfile

Includes manufacturing profiles.

Shrinkage

Includes shrinkage.

Template

Includes templates.

PinJig

Includes pin jigs.

Marking

Includes marking lines.

Weld Compensation

Includes weld compensation.

Enter User Name and Password Dialog Box

Displays the controls used to specify the user name and password under which to run a Manufacturing Service Manager batch job.

Enter the user name

Specifies the user name under which to run the batch job.

Enter the password

Type the password associated with the user name.

Confirm password

Retype the password.

Schedule Wizard - Page 2

Displays the controls used to create, edit, or delete the schedule for a batch job.

Schedule a new job

Creates a new schedule for the selected batch job.

Edit an existing job

Modifies the schedule for the selected batch job.

Delete

Deletes the schedule for the selected batch job.

Schedule Wizard - Page 3

Displays the controls used to specify the frequency at which a batch job is run.

You have selected Batch Name as

Displays the name of the batch job.

Perform this job

Specifies the frequency at which to run the batch job.

Schedule Wizard - Page 4

Displays the controls used to specify the time and day at which a batch job is started.

Start time

Displays the time at which the batch job starts.

Perform this job

Indicates the days on which to run the batch job.

Start date

Displays the date at which to start the schedule.

See Also

Batch Service Manager (on page 266)

MFG Environment Display

Controls what structural manufacturing objects are displayed in the graphic view. This command is located on the **Tools** menu. The following list shows the structural manufacturing objects controlled with this command.

— — **Marking Lines**

 **Fabrication Margin**

 **Position Shrinkage**

 **Scaling Shrinkage**

 **Beveling**

Simple Physical


Turns on and off the simple physical representation of all objects, regardless of whether they have child manufacturing parts.

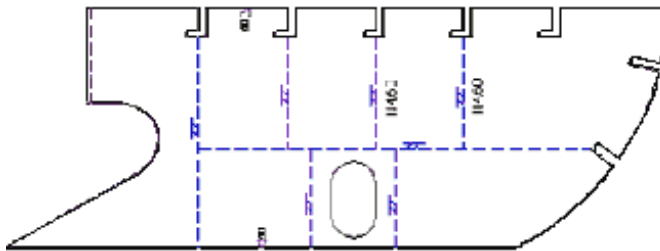
SECTION 12

Part Viewers and Editors

The part viewing and editing commands review and edit parts.

Part Monitor

 Generates and displays XML data for the manufacturing part so that the data can be reviewed before it is output to the nesting or other manufacturing software. The XML can be displayed graphically or as text.



You can monitor the following manufacturing parts that have an XML definition attached to them:

- Plate manufacturing parts
- Profile (stiffener) manufacturing parts

NOTES

- Most markings are automatically applied to the detailed parts by rules. These rules, and the symbols they place, are customizable. For example, you can customize the marking line rules to place marking lines at the left, right, or center of the connected part.
- Marking lines that overlap frame lines are automatically removed from the XML file and part monitor display. The tolerance for the overlap distance is controlled by rules.
- For built-up profiles, the web and flange display on the same sheet in the **Part Monitor**.
- If an object has custom properties, they are included in the generated XML. For example:

```
<SMS_CUSTOM_PROPERTIES INTERFACE_NAME="IJUAAsemblyChild">
<SMS_CUSTOM_PROPERTY NAME="Build Method" VALUE="5" UNITS_TYPE="long"
PRIMARY_UNITS="undefined" CODELIST="BuildMethodData"
CODELIST_VALUE="VerticalDrop"/>
<SMS_CUSTOM_PROPERTY NAME="Slot Connectivity" VALUE="4" UNITS_TYPE="long"
PRIMARY_UNITS="undefined" CODELIST="SlotConnectivityData"
CODELIST_VALUE="Double"/>
</SMS_CUSTOM_PROPERTIES>
```

- Custom XML is created for the following:
 - The part the XML represents
 - The detailed part

- Connected detailed objects
- Holes
- Bevels
- Margins.
- Depending on the type of bevel used in your design, the software changes the bevel symbol used in the **Part Monitor**.

The **Part Monitor** command does not create new objects. It only displays the XML representation of manufacturing objects created by other commands within the Structural Manufacturing task.

Bit Output Options

The following bit output options are available:

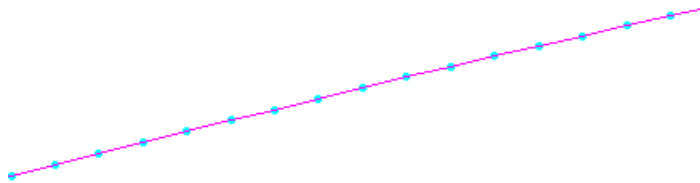
- **XML Purpose**

0 - Basic Output

1 - NC data is added to the XML

2 - Profile sketch

4 - Split curves add an array of points for curve data with higher precision and accuracy controls. The following figure shows a close-up view:



8 - No shrinkage

16 - Support only

32 - Show XML report

64 - Output built-ups as SMS_BUILTUP

- **Annotation Control**

0 - Show nothing

1 - Show annotation only

2 - Show symbols only

4 - Show symbols as specified in **Symbol Control (Long3)**

- **Symbol Control**

1 - Bevel

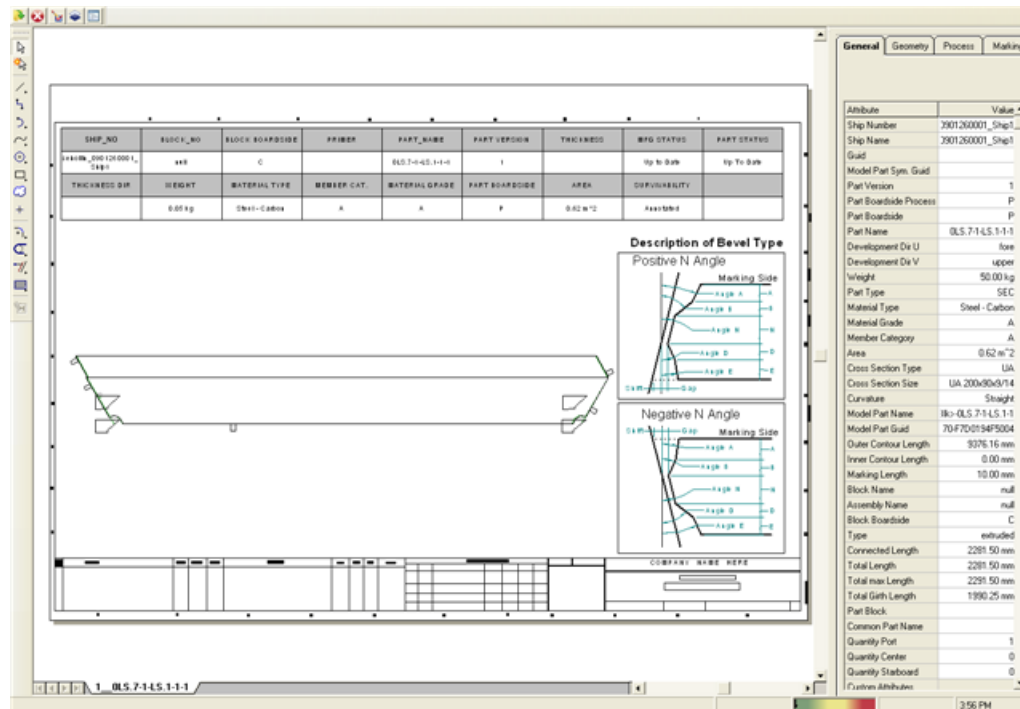
2 - Grind

4 - Knuckle

8 - Part direction

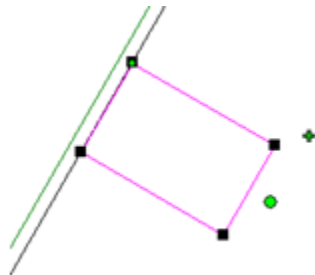
- 16 - Shrinkage
- 32 - Feature symbol
- 64 - Margin
- 128 - Dimensions
- 256 - Labels

The following example shows output using **Plates & Profiles COMBINED** and **Plates & Profiles Symbols Only**.

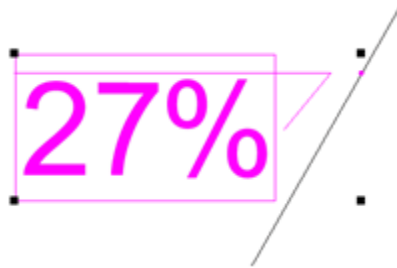


Symbol Controls and **Annotation Controls** have visual differences. **Symbols** load the traditional style of symbols for bevels and other manufacturing attributes. **Annotations** draw everything as groups of 2D objects that then create an annotation entity as defined by the SMS_ANNOTATIONS.xml and the AnnotationSymbols rule.

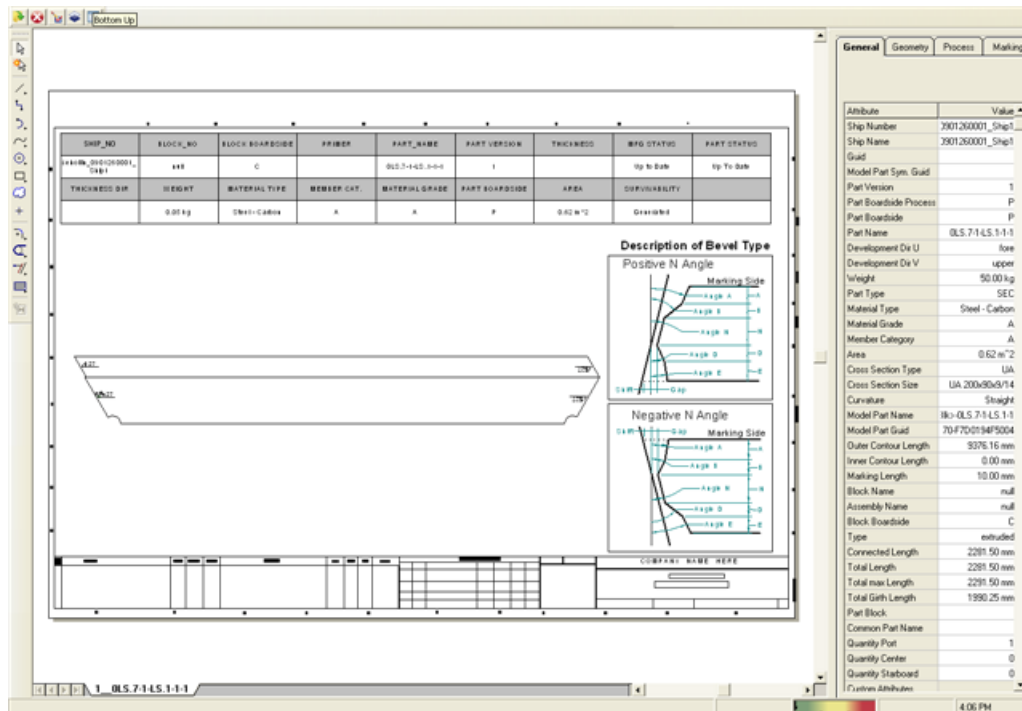
The following example shows a bevel symbol:



The following example shows a bevel annotation:



The following example shows the same profile part with the option set to **Plates & Profiles Annotation**:



- **Viewer Option**
 - 1 - Part Monitor
 - 2 - Part Editor
 - 3 - Annotation Editor
 - 8 - Output Command

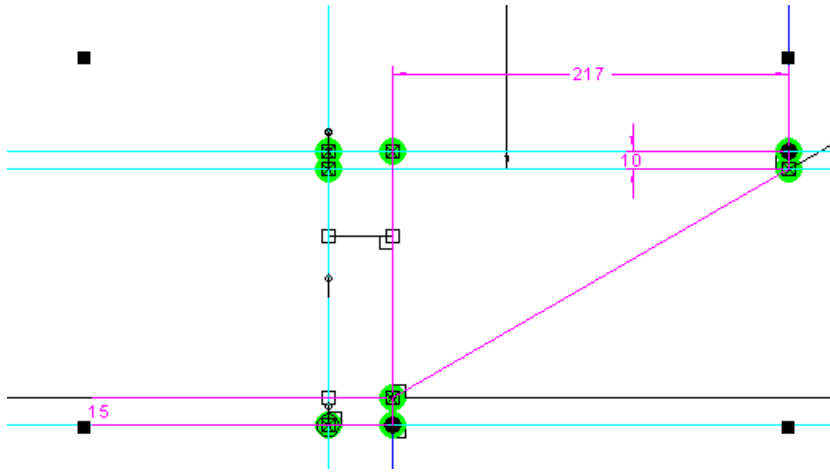
Viewing Dimensions

- With **Symbol Control** set to **128** and **Annotation Control** set to **3** dimensions display in the drawing.
- With **Annotation Control** set to **2**, regardless of what the **Symbol Control** value is set to, dimensions display in the drawing.

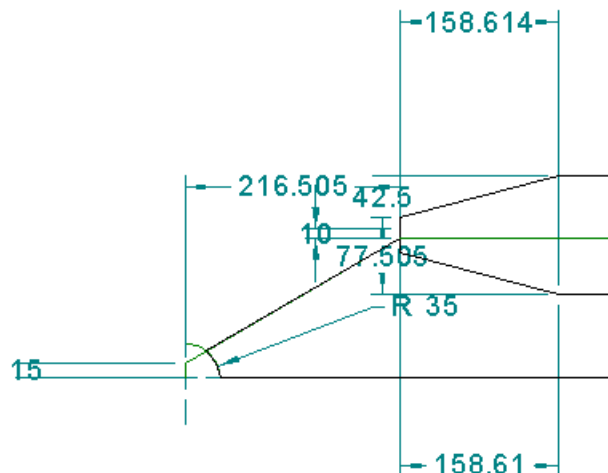
Viewing Dimensions Example

The following example displays a profile output with dimensions.

Symbol output:



Part monitor view:



Manufacturing Plate Display in the Part Monitor

The orientation of manufacturing plates is determined by the software. The logic determines the relationship between the **PlateUpside** (Molded or AntiMolded) property and the **Orientation** rule.

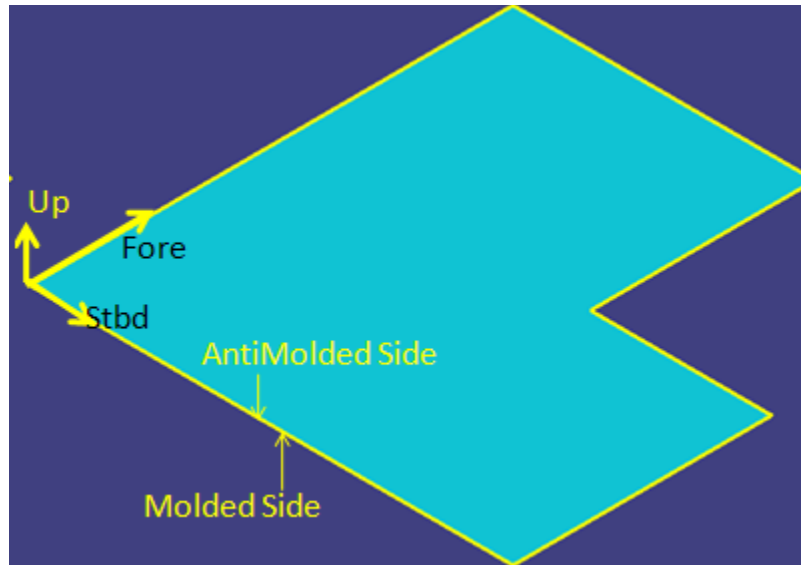
The software makes the following determinations to display the manufacturing plate:

1. The plane (XY, XZ, or YZ) to which the part belongs.

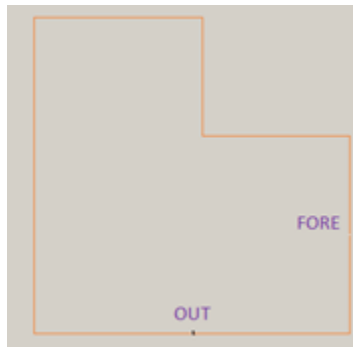
This is the plane in which the part mostly exists. It forms the base plane for the orientation.

- **XY (looking down)** - Parts on the bottom of the shell and on the deck belong to this plane.
- **XZ (looking port)** - Parts on the two sides of the shell and on the longitudinal bulkheads belong to this plane.

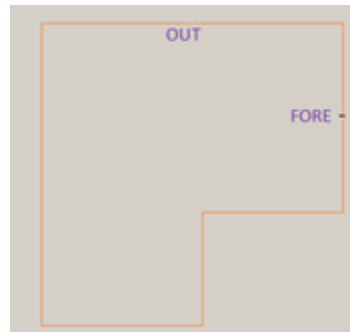
- **YZ (looking forward)** - Parts on the forward or after sides of the shell and on the transverse bulkheads belong to this plane.
2. The orientation in the **Part Monitor**.
- If the part belongs to the XY plane:



This example shows a deck part with a molded side bottom and an anti-molded side up. The manufacturing output is oriented so that fore displays on the right side.

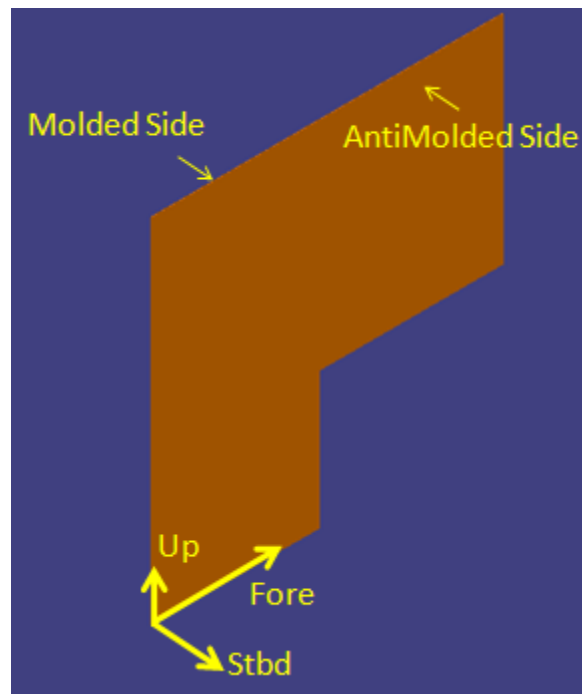


Molded side up

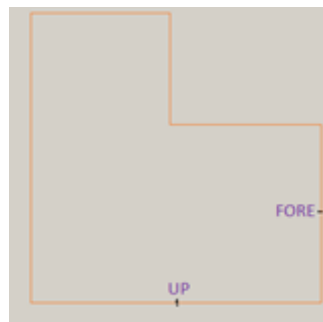


Anti-Molded side up

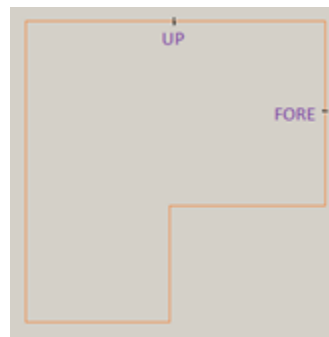
- If the part is in the XZ plane:



The manufacturing output is oriented so that fore displays on the right side.

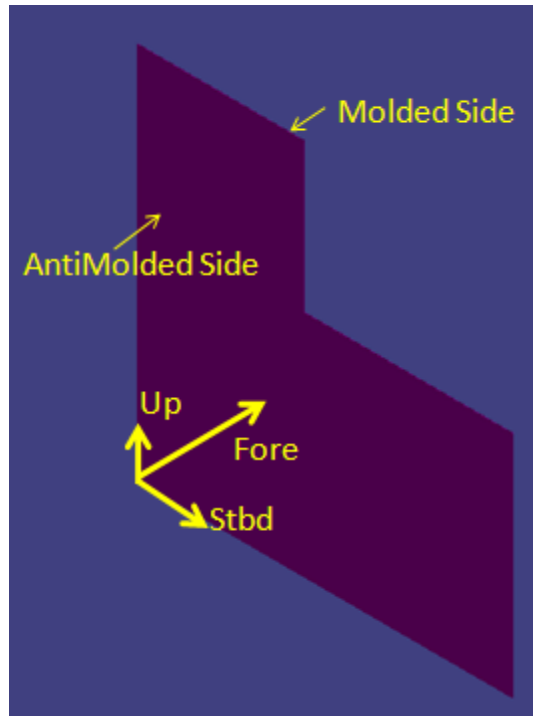


Molded side up

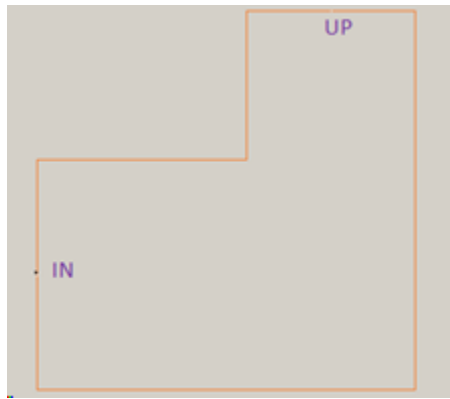


Anti-molded size up

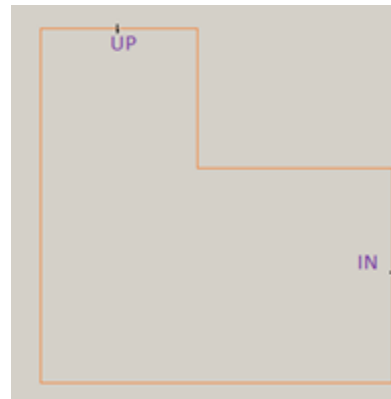
- The part is in the YZ plane:



The manufacturing output is oriented so that up displays on the top.



Molded side up



Anti-molded side up

Part Monitor Ribbon

Displays the controls used to display manufacturing parts.

Select MFG Part

Selects a manufacturing part whose XML data you want to display. You can select the part in a graphic view or in the **Workspace Explorer**.

Open XML File

Displays the **Open** dialog box so that you can select an external XML text file to open.

✓ **Accept**

Opens the specified viewer for the selected parts.

✗ **Reject**

Clears the select set.

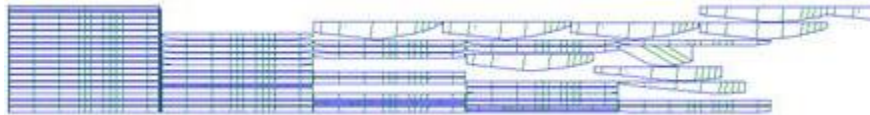
Viewer

Specifies the viewer to use for the selected parts.

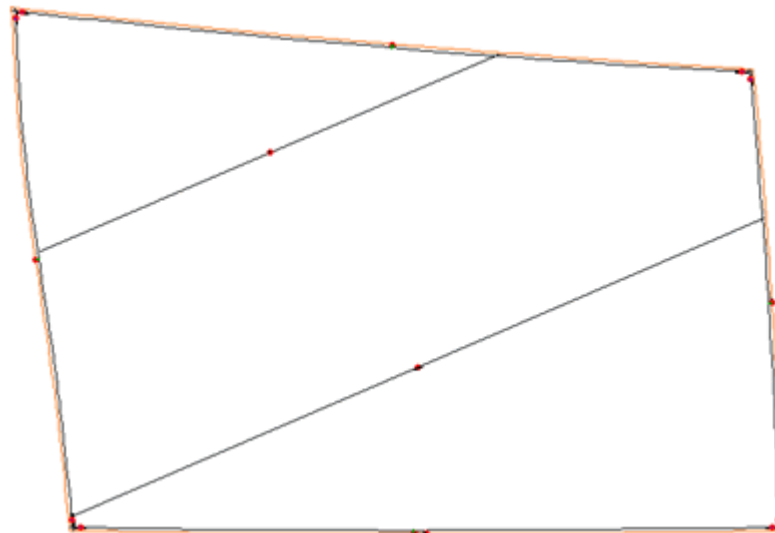
Template

Selects the template that you want to use as a background. Templates are the background files for the **Part Monitor**. Most of them contain text boxes where various commonly-used information is loaded into the 2D view. Some of the template files are empty so that many parts can be loaded, such as the template files used with the various **Check Drawing** modes.

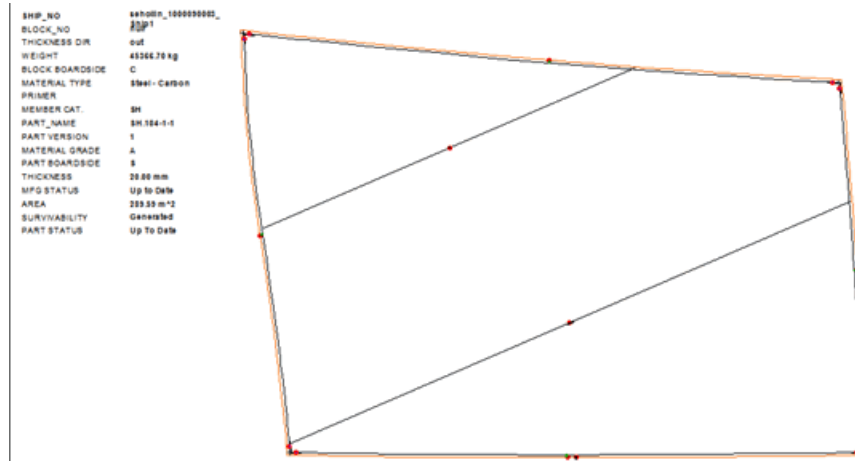
- **Check Drawings** - Reviews a selection of plates within the drawing. This option lets you review the output data as a group rather than individually. The output created by this option is similar to a nesting package. This is not actual nesting software, so the output is not optimized for actual production. When you select **Check Drawings** as the template, the software automatically selects the CheckDrawings.sha template file.



- **Empty** - Uses a blank template. This is the default option.
- **EmptyDefaultXML** - The default template for displaying plates, profiles, or member objects. You can change the documents, but the internal settings must remain.
- **EmptyNoBevel.sha** - Displays output similar to the following figure:



- **EmptyRightSide.sha** - Displays output similar to the following figure:

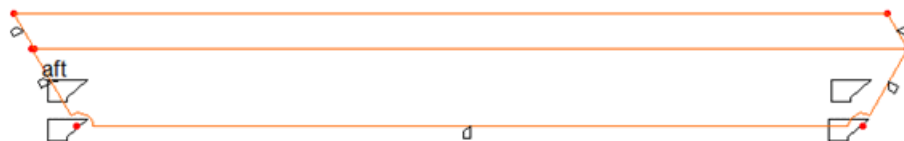


- **MfgTemplateSet** - The default drawing template document for displaying template objects. You can change this document, but the internal settings must remain.
- **PanelDrawings** - Uses a panel template.
- **PanelEmpty** - Uses an empty panel template.

Output Format

Selects the output format rule to use. This affects the filter that is applied as well as many other settings from the data build loaded to the catalog. For example, selecting the **Plates & Profiles, Split Curves** format applies a preprocessing step to the XML output that generates a list of coordinates for curved data and edges. The **Part Monitor** then uses these coordinates to create a spline curve which gives a much cleaner final product.

- **Plates and Profiles COMBINED** and **Plates & Profiles, Symbols** - Uses the traditional symbol engine.



- **Plates and Profiles with Annotation** - Uses a user-defined rule-driven annotation engine. Each annotation is a group of lines, arcs, circles, and text boxes with a set of attributes that comprise the annotation entity.



- **Plates and Profiles as Plates** - Uses rules to show manufacturing as plates.

- **Member SMS_Profiles** - Sets the selector rule for picking members. You can only select members.



- **Plates & Profiles, Split curves** - Supports high quality splines. This option is useful if you must provide output for high-precision cutting robots. The curve is represented by a large number of coordinate vertex points to which a spline can be fitted. The plate part shows a large number of points along the curve.

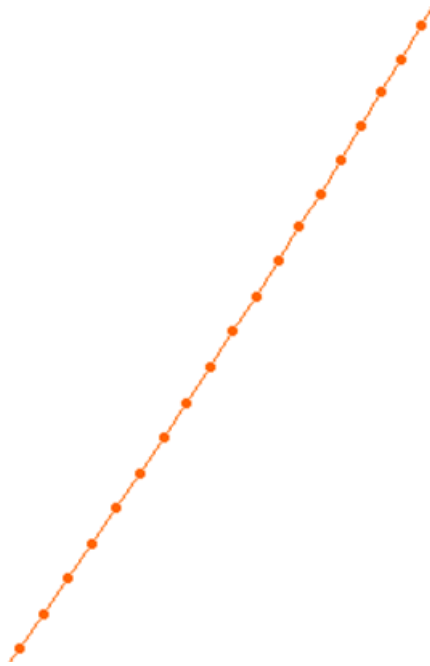
The following image shows the part as it is loaded using either the **Plates & Profiles COMBINED** or **Plates & Profiles, Symbols** options.



The following image shows the part as it is loaded using the **Plates & Profiles, Split Curves** option.



The individual coordinates along the curve can be seen in the following image.



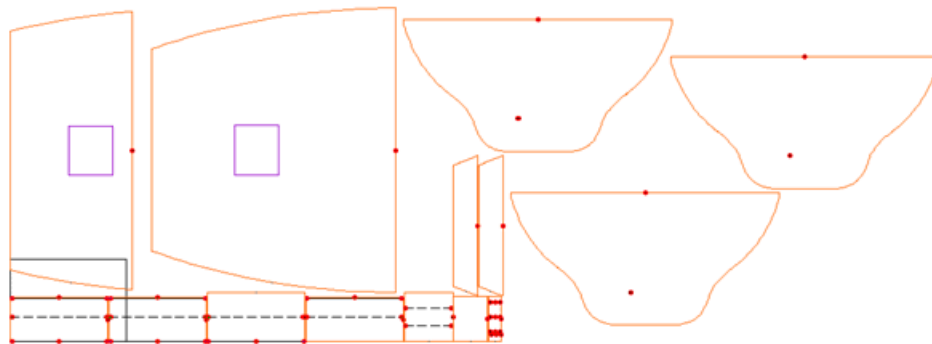
NOTE Parts loaded using the **Plates & Profiles, Split Curves** option take longer to process because of the extra data that must be processed for drawing in the 2D view.

- **Check Drawings SMS_Profiles** - Reviews many parts at once in a format that is similar to a nesting package. This is not nesting software, so the output is not optimized for production. When you select **CheckDrawings SMS_Profiles**, the **Template** box automatically changes to the CheckDrawings.sha template file.

You can locate only profile and member objects when you use this mode.

- **Check Drawings SMS_PLATES** - Reviews many parts at once in a format that is similar to a nesting package. This is not nesting software, so the output is not optimized for production. When you select the **CheckDrawings SMS_PLATES** option, the **Template** box automatically changes to the CheckDrawings.sha template file.

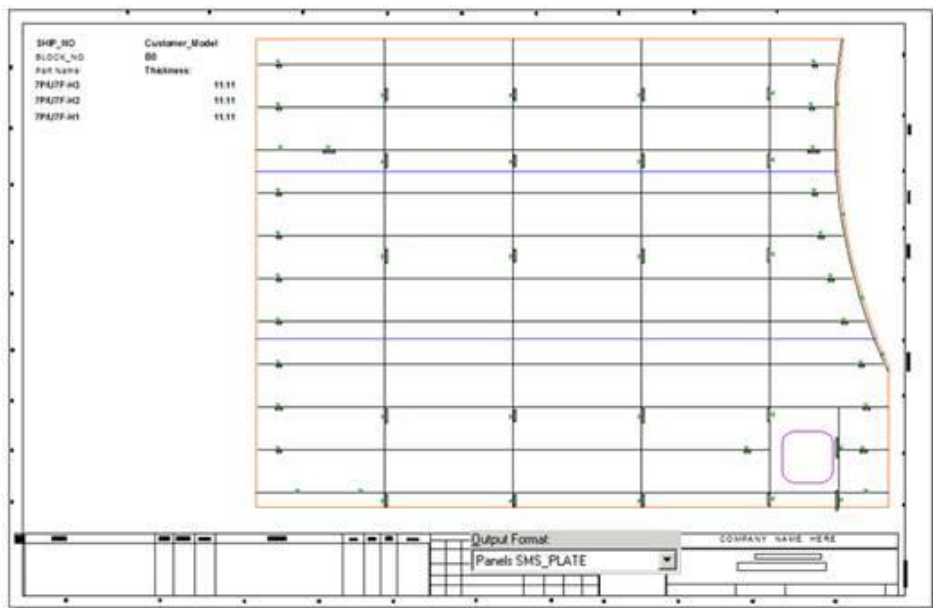
You can locate only plate objects when you use this mode.



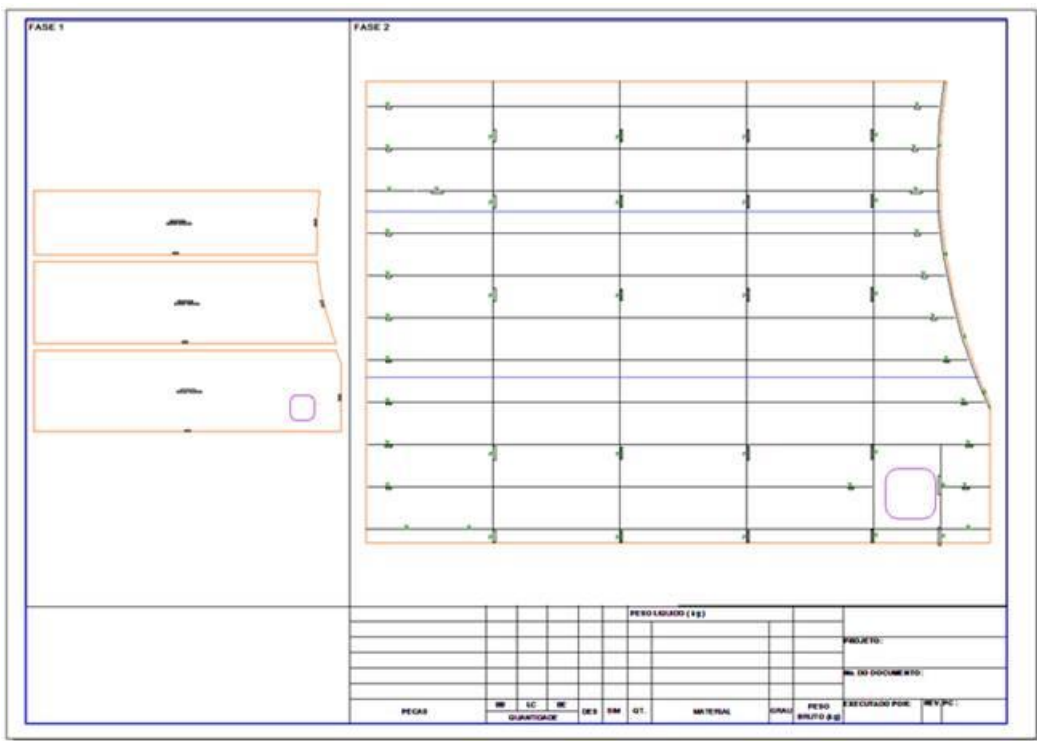
- **Plates & Profiles with Customization** - Creates a plate or profile XML document. This document includes annotations. It is adjusted from the default output using customizable

rules as specified in the catalog table. Common output customization includes the assembly path for plates and cross section information for profiles.

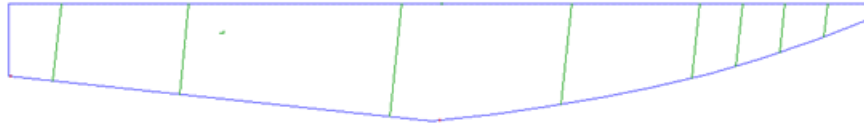
- **Panels SMS_Plate** - Reviews panel objects and displays a report with part information.



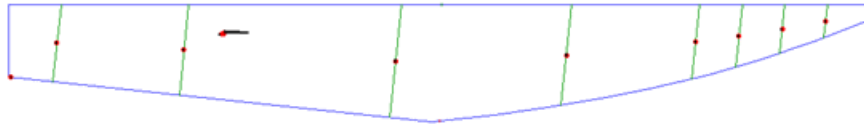
- **Panels - Drawing** - Reviews the panel object and the related parts that make up the panel.



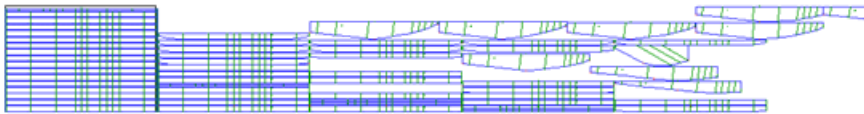
- **Templates SMS_TEMPLATES** - Reviews template sets. The **Template** box automatically changes to the MfgTemplateSet.sha template file.



- **Templates SMS_Plates** - Default drawing template document, used to view template objects based on the export format. You can change this document, but the internal settings must remain.
- **Templates with Annotation** - Reviews template sets. The drawing is created with annotations. For larger numbers of template sets, or template sets with very large numbers of templates, this option can also generate large numbers of annotation entities. This can impact load-time performance. The **Template** box automatically changes to the MfgTemplateSet.sha template file.




- **Templates Check Drawings** - Reviews many templates or template sets at once in a format that is similar to a nesting package. This is not nesting software, so the output is not optimized for production. When you pick the **Check Drawings Templates** option, the **Template** box automatically changes to the CheckDrawings.sha template file.





What do you want to do?

- *Display manufacturing plate XML text* (on page [286](#))
 - *Display manufacturing plate graphically* (on page [287](#))
 - *Display manufacturing profiles XML text* (on page [288](#))
 - *Display manufacturing profiles graphically* (on page [288](#))
 - *Display manufacturing template XML text* (on page [288](#))
 - *Display manufacturing template graphically* (on page [289](#))
-

Display manufacturing plate XML text


1. Click **Part Monitor** .
2. Select **Text** from the **Viewer** list.
3. Select a format, such as Plates SMS_PLATES, from the **Output Format** list.
4. Select the manufactured detailed plates to display from the workspace.

TIPS



- You can select the parts before starting the command.
 - You can select the parts in the model, or click **Open XML File**  to open an external XML file.
5. Click **View** or **Accept** .




Display manufacturing profiles XML text

1. Click **Part Monitor** .
2. Select **Text** from the **Viewer** list.
3. Select a format, such as Profiles SMS_PROFILES, from the **Output Format** list.
4. Select the manufactured detailed profiles to display from the workspace.



TIPS

- You can select the parts before starting the command.
 - You can select the parts in the model, or click **Open XML File**  to open an external XML file.
5. Click **View** or **Accept** .

Display manufacturing profiles graphically


1. Click **Part Monitor** .
2. Select **Drawing** from the **Viewer** list.
3. Select a format, such as Profiles SMS_PROFILES, from the **Output Format** list.
4. Select the manufactured detailed profiles to display from the workspace.

TIPS

- You can select the parts before starting the command.
 - You can select the parts in the model, or click **Open XML File**  to open an external XML file.
5. Click **View** or **Accept** .



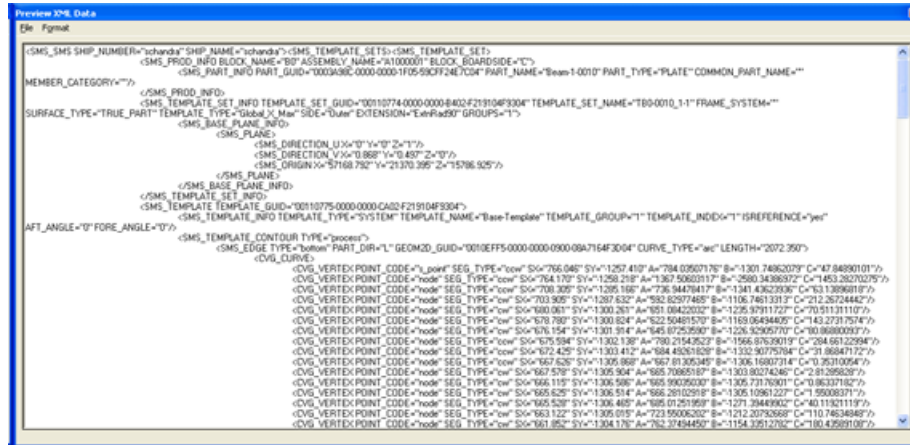
Display manufacturing template XML text

1. Click **Part Monitor** .
2. Select **Text** from the **Viewer** list.
3. Select a format, such as Profiles SMS_TEMPLATES, from the **Output Format** list.
4. Select the part (plate, profile, or member) which has the template to display from the workspace.


TIPS

- You can select the parts before starting the command.



5. Click **View** or **Accept** ✓.

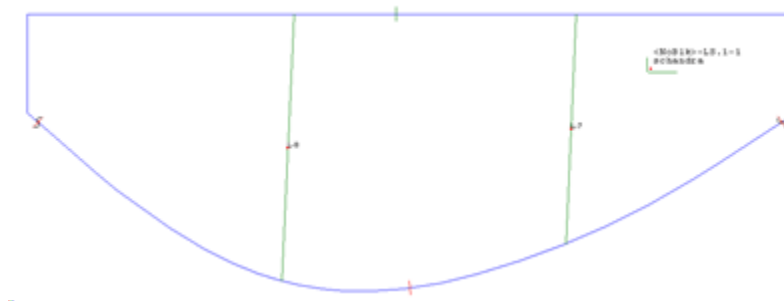


Display manufacturing template graphically

1. Click **Part Monitor** .
2. Select **Drawing** from the **Viewer** list.
3. Select a format, such as Profiles SMS_TEMPLATES, from the **Output Format** list.
4. Select a template, such as MfgTemplateSet.sha, from the **Template** list.
5. Select the part (plate, profile, or member) which has the template to display from the workspace.

TIPS

- You can select the parts before starting the command.
 - You can select the parts in the model, or click **Open XML File**  to open an external XML file.
6. Click **View** or **Accept** .



Part Monitor Dialog Box

Displays the manufacturing part that you have selected. You can use the standard SmartSketch commands to navigate the file, but you cannot make changes to the file.

File Menu

Open

Opens an existing XML file.

Save As

Saves the drawing as an XML file.

Discard Changes and Exit

Returns to the 3D view.

Drawing Ribbon Bar

Close

Returns to the 3D view.

Status

Displays the status of the part.

- **Generated** - All the steps and rules were successfully applied, and the manufacturing part was generated.
- **Annotated** - The annotations on the manufactured part have been edited by the **Annotation Editor**.
- **Edited** - The manufactured part has been edited using the **Part Editor**. The contours, markings, attributes, or annotation on these objects may have been changed.

General tab

Displays the general attributes of the part.

Geometry tab

Displays the information about the selected contour or marking.

Process tab

Displays the process settings values for the part.

Marking tab

Displays the marking settings values for the part.

See Also

Part Monitor (on page 272)

Part Editor



Edits manufactured parts. You can add markings, contours, and annotations to manufactured parts, and then save those changes to the database.

Part Editor Ribbon

Displays the controls used to edit manufacturing parts.



Select Part

Select the part to edit. The value in the **Output Format** box controls the parts that you can select.



Open

Opens a file.



Cancel

Clears the selection.



Accept

Accepts the selection, and starts the **Part Editor**.

Output Format

Select the type of manufacturing part to select in the model. This option sets the filter. You can select only objects in the model that match this type.

What do you want to do?

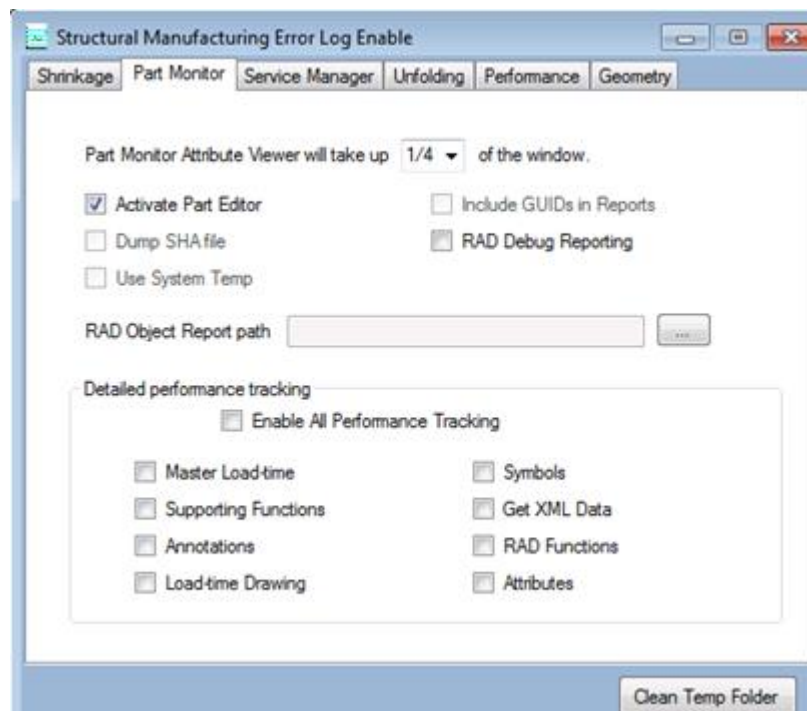
- *Open the Part Editor environment* (on page [292](#))
- *Validate an edited part* (on page 293)
- *Save a modified part to the database* (on page 293)
- *Change the thickness direction of a marking* (on page 293)
- *Change the flange direction of a marking* (on page 293)
- *Change the marking side of a marking* (on page 294)
- *Trim markings* (on page 294)
- *Split markings* (on page 294)
- *Offset a marking* (on page 294)
- *Extend a marking* (on page 295)
- *Trim a contour* (on page 295)
- *Close the corners of a contour* (on page 295)
- *Split a contour* (on page 295)
- *Extend a contour* (on page 295)
- *Zoom in on the features of a contour* (on page 296)

- *Change the size of an annotation* (on page 296)
- *Rotate an annotation* (on page 296)
- *Rotate multiple annotations* (on page 297)
- *Change the attributes of a part* (on page 297)
- *Specify which layers display in the Part Editor* (on page 297)

Open the Part Editor environment

1. Click **Part Editor** on the **Part Monitor** fly-out toolbar.

NOTE If **Part Editor** does not display on the fly-out toolbar, it must first be activated. To activate it, run the executable file *[Product Folder]\StructManufacturing\Client\bin\ManufacturingErrorLogEnable.exe*, and select **Activate Part Editor**.

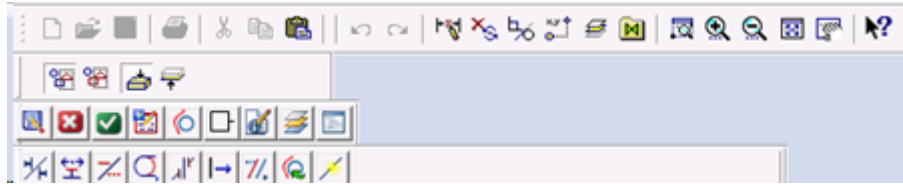


After you restart Smart 3D, or switch to another task and then back to the Structural Manufacturing task, the **Part Editor** is available.

For more information, see *Structural Manufacturing Error Log Enable* (on page 368).

2. Select the manufactured part, and click **Accept** .



The Part Editor opens in 2D.



Save a modified part to the database

1. Click  **Persist to Database**.



Validate an edited part

1. Click  **Validation** to display the **Validation** ribbon.
2. Select the type of validation to run from the **Validation Type** list.
3. Click  **Accept**.

Save a modified part to the database



1. Click  **Persist to Database**.

Change the thickness direction of a marking

1. Click  **Markings** to display the **Markings** ribbon.
2. Select the markings to change.
3. Click  **Toggle Thickness Direction**. You may have to click the command more than once to get to the thickness direction that you need.



The software toggles among the available thickness directions.

Change the flange direction of a marking

1. Click  **Markings** to display the **Markings** ribbon.
2. Select the markings to change.
3. Click  **Toggle Flange Direction**. You may have to click the command more than once to get to the direction that you need.



The software toggles among the available flange directions.

Change the marking side of a marking

1. Click  **Markings** to display the **Markings** ribbon.
2. Select the markings to change.
3. Click  **Toggle Marking Side**.



The software changes the marking to the other marking side, updating the layers and line style.

Trim markings

1. Click  **Markings** to display the **Markings** ribbon.
2. Select the marking to trim.
3. Click  **Trim Marking Line**.





The software trims the selected elements to the nearest intersection.

Split markings

1. Click  **Markings** to display the **Markings** ribbon.
2. Select the marking to split.
3. Click  **Split Marking**.
4. Select the objects by which to split the markings.



The software splits the selected markings at the intersections to the selected objects.

Offset a marking

1. Click  **Markings** to display the **Markings** ribbon.
2. Click  **Offset Object**.
3. Select the objects to offset. You can select individual objects that are not closed or groups of objects that are endpoint connected and not closed.
A temporary object displays in red to show you the position of the copy.
4. If necessary, change the value in the **Offset** box or click  **Toggle Offset Direction** to move the temporary copy to the other side of the original objects.
5. Click  **Accept**.



The software copies the objects to the new location.

Extend a marking

1. Click  **Markings** to display the **Markings** ribbon.
2. Click  **Extend Marking**.
3. Select the object to extend near the end from which you want the extension to start.



The software extends the selected object to the nearest intersection.

Trim a contour

1. Click  **Contours** to display the **Contours** ribbon.
2. Select the contour to trim.
3. Click  **Trim Contour**.



The software trims the selected elements to the nearest intersection.

Close the corners of a contour

1. Click **Contours**  to display the **Contours** ribbon.
2. Click **Close Corner Contour** .
3. Select one of the objects to use to close the contour near the end that you want to extend.
4. Select the other object that to use to close the contour near the end that you want to extend.



The software extends the selected elements to their intersection.

Split a contour

1. Click  **Contours** to display the **Contours** ribbon.
2. Select the contour to split.
3. Click  **Split Contour**.
4. Select the objects by which to split the contour.




The software splits the selected contour at the intersections to the selected objects.

Extend a contour


1. Click  **Contours** to display the **Contours** ribbon.
2. Click  **Extend Contour**.
3. Select an open object to extend near the end from which you want the extension to start.

The software extends the selected object to the nearest intersection.



Zoom in on the features of a contour

1. Click  **Contours** to display the **Contours** ribbon.
2. Select the contour containing the features to examine.
3. Click  **Feature Zoom**.
The software zooms in to the first feature on the selected contour.
4. Click  **Feature Zoom** again to display the next feature on the contour.


Change the size of an annotation

1. Click  **Annotation** to display the **Annotation** ribbon.
2. Select the annotation to change.
3. Click v **Decrease Size** to reduce the size of the annotation, ^ **Increase Size** to enlarge the size of the annotation, or select a size from the **Select Size** list.
The software changes the size of the annotation.


Rotate an annotation


1. Click  **Annotation** to display the **Annotation** ribbon.
2. Select the annotation to rotate.
3. Click  **Rotate**.
*If you have selected a value in the **Select Rotation Angle** box, the software rotates the annotation. Otherwise, the software prompts you for a point of axis for the rotation. Continue on with steps a-c.*
 - a. Click the point in the graphic view to use as the point of axis for the rotation.
The software prompts you for a starting point.
 - b. Click the point in the graphic view to use as the starting point for the rotation.
The software prompts you for an ending point.
 - c. Click the point in the graphic view to use as the ending point for the rotation.
The software rotates the annotation.

Rotate multiple annotations


1. Click **Annotation**  to display the **Annotation** ribbon.
2. Select the annotations to rotate.
3. Select a rotation angle from the **Set Text Rotation** list.

Change the attributes of a part

1. Click  **Attribute Editor** to display the **Part Editor Object Attribute Manager** dialog box.
2. Select an object to view its attributes.
3. Edit the attributes to meet your requirements.
4. Click **Apply**.

 **NOTE** Any changes that you have made to a *contour*, *marking object*, or to a *general part attribute* object are canceled if you click on a new object before you apply the changes.

Specify which layers display in the Part Editor

1. Click  **Layers and Style** to display the **Style** dialog box.
2. Select the check boxes next to the layers to display.

Part Editor Environment

Displays the 2D editing controls as well as the **Part Editor** controls. The **Part Editor** lets you manually edit the manufacturing data for a plate, profile, or member parts by adjusting the contour, deleting contour segments and redrawing them, deleting all of the contours and drawing all new contours, or creating custom cutout features.

Validation

Displays the **Validation** ribbon. For more information, see *Validation Ribbon* (on page 298).

Close

Cancels any changes, exits the **Part Editor**, and returns to the 3D environment.

Persist to Database

Saves the modified part to the database.

Markings

Displays the **Markings** ribbon. For more information, see *Markings Ribbon* (on page 298).

Contours

Displays the **Contours** ribbon. For more information, see *Contours Ribbon* (on page 300).

Annotation

Displays the **Annotation** ribbon. For more information, see *Annotation Ribbon* (on page 301)

**Attribute Editor**

Displays the **Part Editor Object Attribute Manager** dialog box. If you have selected a 2D object, then the dialog box displays the attributes of that object. If you have not selected a 2D object, the dialog box displays general part attributes. If you have selected more than one 2D object, the dialog box does not display. For more information, see *Part Editor Object Attribute Manager Dialog Box* (on page 302).

**Layers and Styles**

Displays the **Style** dialog box. For more information, see *Style Dialog Box* (on page 305).

**Display XML**

Displays the original XML file for the part. If the output format is set to annotation, the annotation XML objects are included.

NOTE Changes to the part are not visible in the XML file viewer for that part until the changes are persisted to the database and the part is reloaded.

Validation Ribbon

Displays settings for running validation of edited manufacturing parts.

**Settings**

Displays the **Validation Settings** dialog box which displays the persistence validation settings. These settings indicate what types of validation are enabled and what level of error halts persistence for each type of validation. These settings are automatically loaded from the configuration file. You cannot modify these settings.

Validation Type

Select the type of validation to run. The list includes the options enabled in the **Part Editor** configuration file.

**Accept**

Accepts the settings and runs the validation. Errors or warnings display on the **Part Validation** dialog box. If the entire text is not visible in a cell, hovering the mouse over that cell displays a tooltip with the complete text. Click **Rerun Validation** on the **Part Validation** dialog box to rerun the previous validation type on the part after you make changes.

Markings Ribbon

Displays controls for adding marking to the manufacturing part. This ribbon displays by default when you open the **Part Editor**. Using these controls, you can adjust marking lines, delete marking lines and redraw them, or delete all the marking and draw all new markings.

Use caution when you use this tool. Persistence must pass a validation phase to ensure consistency such as closed contour segments and attributes that match expected values such as bevel and grind data that total up to the same value as the part thickness.

Validation errors are reported so that you can correct them. If there are only warnings, then persistence does not stop.

- Toggle Thickness Direction

Changes the thickness direction attributes for all selected marking line objects. The `MEASURE_DIR` attribute toggles between **L**, **R**, and **C**. The `THICKNESS_DIR` attribute toggles between the appropriate directions. For example, if the `THICKNESS_DIR` is **fore**, then the software toggles that attribute between **fore**, **aft**, and **centered**. The software also toggles the orientation of any thickness direction symbols associated with the selected objects.

Click **Toggle Thickness Direction** to toggle any selected marking objects. This command remains active until you click the button again or click another command. While this command is active, any marking line that you select has its attributes toggled.

Toggle Flange Direction

Changes the flange direction attributes for all selected marking line objects. The `FLANGE_DIR_2D` attribute toggles between **L** and **R**. The `FLANGE_DIR` attribute toggles between the appropriate directions. For example, the software can toggle the `FLANGE_DIR` attribute between **fore** and **aft**. The software also toggles the thickness direction symbols associated with the selected objects.

Click **Toggle Flange Direction** to toggle any selected marking objects. This command remains active until you click the button again or click another command. While this command is active, any marking line that you select has its attributes toggled.

Toggle Marking Side

Changes the `MARKED_SIDE` attribute for all selected marking line objects from **marking** to **anti_marking**, or the reverse. The software moves objects to the correct layer. For example, objects that were on the **marking** layer before this command are moved to the **anti-marking** layer. The software updates the line style for the changed objects. All objects that are moved to **marking** display in a solid line style. Objects moved to **anti-marking** display in a dashed line style.

Click **Toggle Marking Side** to toggle any selected marking objects. This command remains active until you click the button again or click another command. While this command is active, any marking line that you select has its attributes toggled.

Trim Marking Line

Trims open and closed elements to the closest intersection in both directions. To trim one element at a time, click each element that you want to trim. To trim more than one element at the same time, drag the pointer over the element segments that you want trimmed. When you release the mouse button, all the elements that are touched by the mouse path are trimmed.

Split Marking


Splits the selected marking object.

 **NOTE** You must select at least one object to be split before starting this command.

After you start this command, the software prompts you to select objects to do the splitting. You can select an individual object, or drag the pointer over multiple objects. The attributes on the original objects that you split are inherited by all of the new child objects. It is not possible to split an object by itself.

Offset Object

Creates a copy of a selected line or group of objects at a given distance from the original object along a line normal to the line connecting the endpoints of the object. You can offset

individual objects that are not closed, or groups of objects that are endpoint connected and not closed. A temporary copy of the object displays in red so that you can verify your settings. The temporary copy is not written to the design until you click  **Accept**. This command displays the following options:

 **Accept**

Accepts the settings and creates the offset object.

 **Close**

Cancels any changes, exits the command.

 **Toggle Offset Direction**

Changes the direction of the offset from one side of the original object to the other.

Offset

Specifies the distance that the new object is offset from the original object.

If you click a new object before finalizing the offset, then the original offset is canceled and a new temporary offset is created for the new object.

When you finalize the offset, the software creates the new object with a default marking line attribute set regardless of the type of the original object.

If the original object is a line and its end points are either connected to outer contours or in close proximity to outer contours, then the new offset object is trimmed to the contours after offset.

 **Extend Marking**

Extends open objects to the nearest intersection point starting from the nearest end of the object that you select. If there are no objects within the extension path of the object, then the extend command does not work for that object. To extend one element at a time, click the element that you want to extend nearest to the end point that you want extended.

 **Change to Contour**

Displays the **Contours** ribbon.

Contours Ribbon

Displays options for editing contours.

 **Trim Contour**

Trims open and closed elements to the closest intersection in both directions. To trim one element at a time, click each element that you want to trim. To trim more than one element at the same time, drag the pointer over the element segments that you want trimmed. When you release the mouse button, all the elements that are touched by the mouse path are trimmed.

 **Close Corner Contour**

Connects the end points of two objects. The new intersection point is the mid-point between the two end points before the command is started. It is not possible to close more than one corner contour at a time.

 **Split Contour**

Splits the selected contour object.

 **NOTE** You must select at least one object to be split before starting this command.

After you start this command, the software prompts you to select objects to do the splitting. You can select an individual object, or drag the pointer over multiple objects. The attributes on the original objects that you split are inherited by all of the new child objects. It is not possible to split an object by itself.

Extend Contour

Extends open objects to the nearest intersection point starting from the nearest end of the object that you select. If there are no objects within the extension path of the object, then the extend command does not work for that object. To extend one element at a time, click the element that you want to extend nearest to the end point you want extended.

Change to Marking

Deletes the attributes from all selected contour objects, and adds a default set of marking attributes to those objects. This command gives a default set of marking attributes to all of the selected objects that lack attributes and that are not located on the text or symbol layers. This command deletes any contour symbols associated with the selected objects. Every object changed to a marking object is placed on the marking layer. The displayed ribbon switches from the **Contours** ribbon to the **Markings** ribbon.

Feature Zoom

Automatically zooms to each feature on the part. Click the command again to automatically zoom to the next feature on the part if there is more than one feature. If the part has many features, then clicking the command sequentially and automatically zooms to each of the features. When all the features have been displayed, the command returns to the first feature and continues.

Annotation Ribbon

Displays controls for adding and editing annotation objects.

Decrease Size

Reduces the size of the annotation. The available sizes are controlled through the SMS_ANNOTATION.xml file.

Increase Size

Enlarges the size of the annotation. The available sizes are controlled through the SMS_ANNOTATION.xml file.

Select Size

Select a size for the annotation from the list. The available sizes are controlled through the SMS_ANNOTATION.xml file. The values depend on the type of object that you have selected to edit. For example, if you have selected a plate part, the list contains those values specified for plates.

Rotate

Rotates the selected annotation. The software prompts you to select a rotation axis point, a rotation starting point, and a rotation end point. The rotation end point is not needed if you select a value in the **Select Rotation Angle** box.

Select Rotation Angle

Select a rotation angle for the annotation. The available angles are controlled through the SMS_ANNOTATION.xml file. If you select a value in this box, the software uses that angle to compute the rotation end point.

```
<ROTATION_CONTROL>
  <ROTATION_SETTING NAME="angle" VALUE="0" ALLOWABLE="-90,-45,-30,-
15,15,30,45,90"/>
</ROTATION_CONTROL>
```

Type

Select the annotation type to place. The list contains the annotation types included in the SMS_ANNOTATION.xml file using the attribute name <TYPE> and <DISPLAY_NAME>.

```
<SMS_OUTPUT_ANNOTATION TYPE="FURIWAKEMARK"
  DISPLAY_NAME="FuriwakeThickness1" ... >
<SMS_OUTPUT_ANNOTATION TYPE="GRINDEND" DISPLAY_NAME="Grind End"
  ... >
<SMS_OUTPUT_ANNOTATION TYPE="GRINDEND2" DISPLAY_NAME="Grind
  Start" ... >
```

Make

Displays the **Part Editor Object Attribute Manager** dialog box which creates custom annotation for the selected object. For more information, see *Part Editor Object Attribute Manager Dialog Box* (on page 302). The custom annotation attributes such as text and geometry are controlled by the SMS_ANNOTATION.xml file. Making an annotation adds the following:

- **Annotation** attribute set to the selected group.
- **Annotation_Text** attribute set to all text inside the group. For each text inside the group, a *CustomText_<number>* is added as an attribute.
- **Annotation_Edge** attribute set to all edges inside the group.

Part Editor Object Attribute Manager Dialog Box

Displays options to control general part attributes. If no object is selected, the dialog box displays general part attributes. If a 2D object is selected, the dialog box displays the attributes for that object. If you have selected more than two 2D objects, the dialog box is not available.

If the dialog box cannot find the SMS_VIEWER.xml configuration file, it loads in read-only mode. This file is delivered to the *[Product Folder]\SharedContent\StructManufacturing\SMS_SCHEMA* folder.

While the dialog box is open, you can select different objects to display their attributes.

NOTE Any changes that you have made to a *contour*, *marking*, or *general* object are canceled if you click on a new object before you apply the changes.

The values associated with the attributes displayed on this dialog box fall into the following categories:

- **Text** -Type any string of normal text.
- **Unit** - Type a decimal number followed by a unit. If you type only a number, the software applies the default unit for that box.
- **List** - Select a value from the list.

- **Read-only** - You cannot edit read-only boxes.

Object Type

Displays the type (**Marking**, **Contour**, or **General**) of the currently selected object.

Bevel Category

Displays the basic bevel type (such as **I**, **K**, or **Y**) associated with the currently selected contour. This option is only available when you select a contour with a defined bevel.

Bevel Sub-category

Displays the specific bevel within a class, such as the one with a 20-degree bevel angle and a 2-mm root gap. This option is only available when you select a contour with a defined bevel. The default subtype is **Auto**, which applies a subtype based on the thickness of the part.

Grind Category

Displays the basic grind type (such as **I**, **K**, or **Y**) associated with the currently selected contour. This option is only available when you select a contour with a defined grind.

Grind Sub-category

Displays the specific grind within a class, such as the one with a 20-degree bevel angle and a 2-mm root gap. This option is only available when you select a contour with a defined grind. The default subtype is **Auto**, which applies a subtype based on the thickness of the part.

General Attribute Guidelines

- This dialog box uses the general attribute **SMS_PLATE||THICKNESS** to calculate predefined bevels on contours. Use caution when modifying this attribute.

Contour Attribute Guidelines

- The **SMS_EDGE||TYPE** attribute is the most powerful attribute for a contour. If the **SMS_EDGE||TYPE** attribute is changed from one type to another, then attributes that are not defined for the new **SMS_EDGE||TYPE** are discarded, and attributes defined for the new **SMS_EDGE||TYPE** but not defined for the old **SMS_EDGE||TYPE** are added to the attribute set. Attributes that are common to both the old and the new **SMS_EDGE||TYPE** are preserved without change.
- The **SMS_BEVEL||TYPE**, **SMS_GRIND||TYPE**, and **SMS_PART_MARGIN_INFO||MARGIN_TYPE** attributes similarly act as control attributes. If those attributes are set to "(null)" then the bevel, margin, and grind attributes are not present on the object or in the display. However, setting **SMS_BEVEL||TYPE**, **SMS_GRIND||TYPE**, and **SMS_PART_MARGIN_INFO||MARGIN_TYPE** to something other than null adds bevel, grind, or margin attributes onto the object.
- If a contour has bevel or grind attributes, the following attributes display:

<attribute> Type - Specifies the basic class of bevel or grind, such as **I**, **K**, or **Y**.

<attribute> **Sub-Category** - Specifies the specific bevel or grind within a class, such as a 20 degree bevel angle and a 2 mm root gap.

Bevel and grind types and subtypes are enumerated in the Part Editor default settings file and are defined in a custom VisualBasic project. For more information, see

PREDEFINED_BEVEL_GRIND_CONTROL in the *Structural Manufacturing Reference Data Guide*, or contact your product support specialist.

When you select a bevel or grind type, the editor automatically applies the **Auto** subtype. This applies a bevel or grind based on the part thickness and possibly other attributes. You can override the **Auto** subtype by selecting another setting from the list.

- Whenever the bevel, grind or margin attributes are modified, the bevel, grind, and margin symbols as well as any associated text boxes, are automatically updated.

Marking Attribute Guidelines

- The **SMS_MARKING||TYPE** attribute is the most powerful attribute for a marking. If the **SMS_MARKING||TYPE** attribute is changed from one type to another, then attributes that are not defined for the new **SMS_MARKING||TYPE** are discarded, and attributes defined for the new **SMS_MARKING||TYPE**, but not defined for the old **SMS_MARKING||TYPE**, are added to the attribute set. Attributes that are common to both the old and the new **SMS_MARKING||TYPE** are preserved without change.

For example, by default, the `strmfgr_platelocation_mark` **SMS_MARKING||TYPE** contains the **SMS_MARKING_PART_LOCATION||LOCATED_PART_NAME**, **SMS_MARKING_PART_LOCATION||THICKNESS_DIR**, **SMS_EDGE||PART_DIR**, **SMS_MARKING_PART_LOCATION||DECLIVITY**, and **SMS_MARKING_PART_LOCATION||MEASURE_DIR** attributes, but the `strmfgr_general_mark` **SMS_MARKING||TYPE** does not. Switching from `strmfgr_platelocation_mark` to `strmfgr_general_mark` causes those attributes to be discarded.

Annotation Attribute Guidelines

- Annotations can only be edited when the **Annotation** ribbon is active. For more information, see *Annotation Ribbon* (on page 301).
- Annotations are edited by changing the values of text attributes using the **Annotation** ribbon. When you click **Apply**, a new annotation is created with the updated attribute values and the selected annotation is replaced with the updated one.
- Multiple annotations can be placed by selecting multiple objects of the same type.
- Annotation properties inherit their properties from the currently selected object if the attributes from the currently selected object are compatible with the attributes from the current annotation type.
- The geometry attributes can be edited by changing the `SMS_ANNOTATION.xml` file. Do not edit the geometry attribute names in the `SMS_ANNOTATION.xml` file for existing annotations. The names of the geometry attributes are used in the annotation rules, and changing the names can cause the rules to stop working.
- Annotations can also be edited by changing a selected attribute's values directly in the **Attribute Editor**. You can also select an annotation object while the **Attribute Editor** is open. Click **OK** to regenerate the annotation object with the new values, update the attribute sets accordingly, and close the **Attribute Editor**. Click **Apply** to regenerate the annotation object

with the new values and update the attribute sets, without closing the **Attribute Editor**. You can then make additional changes, click on another object to make additional edits, or click **Cancel** to cancel any unapplied changes.

Style Dialog Box

Displays options to control visual settings in the **Part Editor**. Specifically, this dialog box controls which layers are displayed.

Layers Tab (Style Dialog Box)

Displays options to control which layers display.

All Contours

Turns the display of all the contour layers on or off.

- **Contours**
- **Openings**
- **Features**
- **Points**

All Markings

Turns the display of all the marking layers on or off.

- **Markings**
- **Anti-Markings**
- **Points**

All Annotations

Turns the display of all the annotation layers on or off.

- **Annotations/Symbols**
- **Bevel Annotations/Symbols**
- **Annotation Text/Labels**
- **Points**

All References

Turns the display of all the reference layers on or off.

- **Reference/Symbols**
- **Dimensions**


Colors Tab (Style Dialog Box)

This tab is not currently available.

Text Tab (Style Dialog Box)

This tab is not currently available.

Annotation Editor

 Adjusts annotation objects and saves those changes back to the manufacturing plate, profile, or member part. Use this command to adjust annotations when the plate geometry is correct but you need to change the annotations. This command lets you move, rotate, and scale annotations to avoid overlap, delete unnecessary annotations, and add annotations for clarity during production. You can edit and regenerate annotations by changing annotation attributes and annotation drawing geometry inputs. Annotations can also be deleted.

Bulk Editing

You can place multiple annotations at once if the selected objects are all of the same type with the same attribute sets. Annotations can also be bulk scaled or bulk rotated, allowing for faster and more efficient workflows.

Basic XML Structure

The root node of the configuration file is <SMS_OUTPUT_ANNOTATION>. At the highest level, the configuration file is divided into categories for each type of annotation output.

<OUTPUT_ANNOTATION> contains the settings for the text arguments and geometry arguments used to construct the annotation object.

Each annotation type XML node looks similar to the following figure:

(VB6)


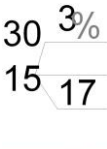




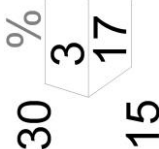

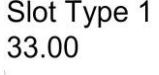



```
<SMS_OUTPUT_ANNOTATION TYPE="ENTITYNAME" DISPLAY_NAME="Entity Name" PROGID="AnnotationSymbols.EntityName" CONTEXT="Marking">
  <SMS_PLACEMENT>
    <INCLUDE_CONDITION NAME="ANGLE_000_LOCAL"/>
  </SMS_PLACEMENT>
  <SMS_INPUT_ARGS>
    <SMS_TEXT_ARGS>
      <INCLUDE_PROPERTY NAME="SMS_ANNOTATION||GEOM2D_GUID"/>
      <INCLUDE_GROUP NAME="PART_FACE"/>
      <INCLUDE_PROPERTY NAME="SMS_ANNOTATION||MARKING_NAME"/>
      <INCLUDE_PROPERTY NAME="SMS_ANNOTATION||REFERENCE"/>
    </SMS_TEXT_ARGS>
    <SMS_GEOM_ARGS>
      <SMS_GEOM_ARG NAME="ControlPoint" VALUE="1m"/>
      <SMS_GEOM_ARG NAME="TextSize" VALUE="30" READONLY="1"/>
      <SMS_GEOM_ARG NAME="TextFont" VALUE="Arial"/>
      <SMS_GEOM_ARG NAME="TextStyle" VALUE="italic"/>
      <SMS_GEOM_ARG NAME="HorizontalOffset" VALUE="0" READONLY="1"/>
      <SMS_GEOM_ARG NAME="VerticalOffset" VALUE="0" READONLY="1"/>
    </SMS_GEOM_ARGS>
  </SMS_INPUT_ARGS>
</SMS_OUTPUT_ANNOTATION>
```


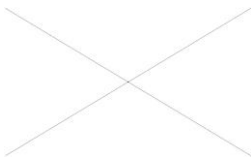
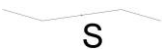
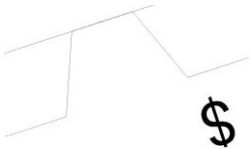




(.NET)















```
<SMS_OUTPUT_ANNOTATION TYPE="ENTITYNAME" DISPLAY_NAME="Entity Name" PROGID="AnnotationSymbolsRule.Ingr.SP3D.Content.Manufacturing.EntityName"
CONTEXT="Marking">
  <SMS_PLACEMENT>
    <INCLUDE_CONDITION NAME="ANGLE_000_LOCAL"/>
  </SMS_PLACEMENT>
  <SMS_INPUT_ARGS>
    <SMS_TEXT_ARGS>
      <INCLUDE_PROPERTY NAME="SMS_ANNOTATION||GEOM2D_GUID"/>
      <INCLUDE_GROUP NAME="PART_FACE"/>
      <INCLUDE_PROPERTY NAME="SMS_ANNOTATION||MARKING_NAME"/>
      <INCLUDE_PROPERTY NAME="SMS_ANNOTATION||REFERENCE"/>
    </SMS_TEXT_ARGS>
    <SMS_GEOM_ARGS>
      <SMS_GEOM_ARG NAME="ControlPoint" VALUE="1m"/>
      <SMS_GEOM_ARG NAME="TextSize" VALUE="30" READONLY="1"/>
      <SMS_GEOM_ARG NAME="TextFont" VALUE="Arial"/>
      <SMS_GEOM_ARG NAME="TextStyle" VALUE="italic"/>
      <SMS_GEOM_ARG NAME="HorizontalOffset" VALUE="0" READONLY="1"/>
      <SMS_GEOM_ARG NAME="VerticalOffset" VALUE="0" READONLY="1"/>
    </SMS_GEOM_ARGS>
  </SMS_INPUT_ARGS>
</SMS_OUTPUT_ANNOTATION>
```

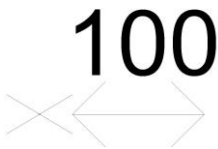

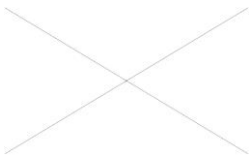


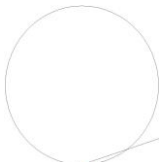
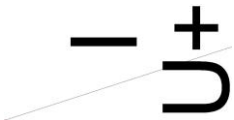

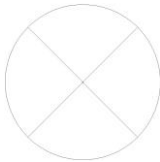

Delivered Annotation Types

You can create new annotations of any type. The following list describes the annotation types delivered with the software.

Annotation Type	Example	Annotation Type	Example
Arrow		Bevel-Parallel	
Arrow-Prefix		Car Deck	
Bevel Change		Knuckle Line	
Bevel-Normal		Centerline	
Collar Plate Info		Lap Connection-Plate	
Conn Part Info-Both Sides		Lap Connection-Profile	

Conn Part Info-One Side		Lap Trace	Marking Name 1
Corrugate Reference		Margin-Fixed	
Corrugate		Mount Angle	\$22.00
End Fitting-Bracket		Outfitting	
End Fitting-End		Plate Id	I +U MA 10 A 0.00 A G
End Fitting-Start		Plate Id-Inverted	I U+ MA 10 A 0.00 A G
Entity Name	Marking Name 1	Profile-Id	I Prof-1.1 A

Flange Direction		Punching	
Grinder		Rend-Circular	
Grinder-End		Rend-Linear	
Grinder-Start		Roll Boundary	
Handle		Roll Line	
Inverse Bending Interval		Rope	
Seam Control-Diamond		Template Half Pipe	

Seam Control-Displacement		Template Knuckle	
Seam Control-X		Template Profile HRB	
Ship Direction		Template Profile RB	
Ship Direction -Inverted		Template Rend	
Sightline		Template Seam	
Special			
Tack Weld			

Annotation Editor Ribbon

Displays options for selecting parts to edit the annotations.

Select Part

Select the part for which to edit the annotation. You can only select parts compatible with the selection in the **Output Format** box.

Open

Opens the **Annotation Editor** from an XML file that you select.

Accept

Accepts the selected objects and opens the **Annotation Editor** environment.

Cancel

Cancels the object selection.

Output Format

Specifies the selection filter. Select the option that matches what you want to view. The options available in this list depend upon the selection rules in the catalog. The selection in this box limits the parts that you can select.

Annotation Editor Environment

Displays options for editing annotations.

Close

Cancels any changes, exits the **Annotation Editor**, and returns to the 3D environment.

Persist to Database

Saves the annotation changes to the database. This command *does not* write any changes to geometry to the database. Only the annotation objects are persisted.

Annotation

Displays the **Annotation** ribbon. For more information, see *Annotation Ribbon* (on page 301).

Attribute Editor

Displays the **Part Editor Object Attribute Manager** dialog box. If you have selected a 2D object, then the dialog box displays the attributes of that object. If you have not selected a 2D object, the dialog box displays general part attributes. If you have selected more than one 2D object, the dialog box does not display. For more information, see *Part Editor Object Attribute Manager Dialog Box* (on page 302).

Layers and Styles

Displays the **Style** dialog box. For more information, see *Style Dialog Box* (on page 305).

Display XML

Displays the original XML file for the part. If the output format is set to annotation, then the annotation XML objects are included.

 **NOTE** Changes to the part are not visible in the XML file viewer for that part until the

changes are persisted to the database and the part is reloaded.

Annotation Control

You can control the size of the selected annotation by using the <PART_EDITOR_CONTROL> in the SMS_Annotation.xml file.

```
<FONT_CONTROL PURPOSE="PROFILE">
  <FONT_SETTING NAME="size" VALUE="30" ALLOWABLE="21"/>
</FONT_CONTROL>
<FONT_CONTROL PURPOSE="PLATE">
  <FONT_SETTING NAME="size" VALUE="40" ALLOWABLE="50,40,30,20,10"/>
</FONT_CONTROL>
```

Decrease Size

Reduces the size of the annotation. The available sizes are controlled through the SMS_ANNOTATION.xml file.

Increase Size

Enlarges the size of the annotation. The available sizes are controlled through the SMS_ANNOTATION.xml file.

Select Size

Select a size for the annotation from the list. The available sizes are controlled through the SMS_ANNOTATION.xml file. The values depend on the type of object that you have selected to edit. For example, if you have selected a plate part, the list contains those values specified for plates.

Rotate

Rotates the selected annotation. The software prompts you to select a rotation axis point, a rotation starting point, and a rotation end point. The rotation end point is not needed if you select a value in the **Select Rotation Angle** box.

Select Rotation Angle

Select a rotation angle for the annotation. The available angles are controlled through the SMS_ANNOTATION.xml file. If you select a value in this box, the software uses that angle to compute the rotation end point.

```
<ROTATION_CONTROL>
  <ROTATION_SETTING NAME="angle" VALUE="0" ALLOWABLE="-90,-45,-30,-
15,15,30,45,90"/>
</ROTATION_CONTROL>
```

Type

Select the annotation type to place. The list contains the annotation types included in the SMS_ANNOTATION.xml file using the attribute name <TYPE> and <DISPLAY_NAME>.

```
<SMS_OUTPUT_ANNOTATION TYPE="FURIWAKEMARK"
  DISPLAY_NAME="FuriwakeThickness1" ... >
<SMS_OUTPUT_ANNOTATION TYPE="GRINDEND" DISPLAY_NAME="Grind End"
  ... >
<SMS_OUTPUT_ANNOTATION TYPE="GRINDEND2" DISPLAY_NAME="Grind
  Start" ... >
```

What do you want to do?


- *Create an annotation* (on page 313)
 - *Save annotation changes to the database* (on page 313)
 - *Edit the annotation for a part* (on page 314)
 - *Increase the size of annotation* (on page 314)
 - *Decrease the size of annotation* (on page 314)
 - *Rotate annotation* (on page 315)
-

Create an annotation

1. Click **Annotation Editor** .

*The **Annotation Editor Ribbon** displays.*


2. To place a geometry specific annotation, select the 2D geometry objects. For example, select a marking line or contour. To place a generic annotation such as plate identification, do not select any 2D geometry.

 **TIP** You can select multiple 2D geometry objects to place multiple annotations. The 2D geometry objects should be of the same type and should have very similar attributes. Attributes that are not the same are not correctly inherited by the annotation.

3. Select the annotation type to place from the **Type** box.

*The **Place Annotation** dialog box displays.*

4. Set up the attribute values to meet your needs.

 **TIP** The annotation attributes are inherited from the currently selected 2D object. You must edit the values manually before placement. If the attribute values are different, they do not display at placement.




5. Click **OK**.

The software creates the annotations on the selected objects.






Save annotation changes to the database

1. Click  **Persist to Database**.





Edit the annotation for a part

1. Click **Annotation Editor** .
*The **Annotation Editor Ribbon** displays.*
2. Select the part type for which to edit the annotation from the **Output Format** list.
3. Select the part for which to edit the annotation.
4. Click **Accept** .
*The **Annotation Editor Environment** displays.*
5. Use the options available in the **Annotation Editor Environment** to edit the annotation.
6. Click **Persist to Database**  to save your changes.

Increase the size of annotation

1. Click **Annotation Editor** .
*The **Annotation Editor Ribbon** displays.*
2. Select the part type for which to edit the annotation from the **Output Format** list.
3. Select the part for which to edit the annotation.
4. Click **Accept** .
*The **Annotation Editor Environment** displays.*
5. Select the annotations to make larger.
 **TIP** You can select multiple annotations and increase the font size for all of them at the same time.
6. Click **Increase Size** , or select a larger size from the **Select Size** list.
7. Click **Persist to Database**  to save your changes.

Decrease the size of annotation


1. Click **Annotation Editor** .
*The **Annotation Editor Ribbon** displays.*
2. Select the part type for which to edit the annotation from the **Output Format** list.
3. Select the part for which to edit the annotation.
4. Click **Accept** .
*The **Annotation Editor Environment** displays.*
5. Select the annotations to make smaller.
 **TIP** You can select multiple annotations and decrease the font size for all of them at the same time.
6. Click **Decrease Size** , or select a smaller size from the **Select Size** list.

- Click **Persist to Database**  to save your changes.

Rotate annotation


- Click **Annotation Editor** .

*The **Annotation Editor Ribbon** displays.*

- Select the part type for which to edit the annotation from the **Output Format** list.
- Select the part for which to edit the annotation.
- Click **Accept** .

*The **Annotation Editor Environment** displays.*

- Select the annotations to rotate.

 **TIP** You can select multiple annotations and rotate them at the same time by using the **Set Text Rotation** option.

- Click **Rotate** .

*If you have selected a value in the **Select Rotation Angle** box, the software rotates the annotation. Otherwise, the software prompts you for a point of axis for the rotation. Continue on with steps a-c.*

- Click the point in the graphic view to use as the point of axis for the rotation.

The software prompts you for a starting point.

- Click the point in the graphic view to use as the starting point for the rotation.

The software prompts you for an ending point.

- Click the point in the graphic view to use as the ending point for the rotation.

The software rotates the annotation.

- Click **Persist to Database**  to save your changes.

Place Annotation Dialog Box

Displays options for editing annotations.

Attribute

Displays the name of the attribute.

Value

Displays the value associated with the attribute. Values that are read-only display in a gray box. Read-only attributes are controlled by the SMS_ANNOTATION.xml file.

NOTES

- The properties defined in the SMS_PROPERTIES.xml file are reused in the SMS_VIEWER.xml file. Because of this, changes in the SMS_PROPERTIES.xml file must be made with caution as they can have other consequences in the software. For more information, see Customizing the Part Editor in the *Structural Manufacturing Reference Data Guide*.

- You can place annotations on multiple contour primitives with a single placement action. The selected geometries must have the same attributes, other than their GUID, or the attributes are not inherited by the annotations.

SECTION 13

Manufacturing Service Manager



Manufacturing Service Manager (in the Structural Manufacturing task) or **Tools > Manufacturing Service Manager** reviews and updates manufacturing objects. **Review** displays a report of the current condition of the manufacturing objects that you have selected. **Updating** pulls any outstanding changes into the manufacturing environment. The command is available on the vertical toolbar in the Structural Manufacturing task, and by clicking **Tools > Manufacturing Service Manager** in the Molded Forms, Structural Detailing, Planning, and Structural Manufacturing tasks.

You can select individual parts, assemblies, or blocks for this command.

The **Manufacturing Service Manager** pulls changes into the manufacturing environment instead of associatively pushing from other tasks into the manufacturing environment. In this way, the manufacturing environment is similar to the drawing environment; you have control over which objects get updated and when.

NOTES

- A manufacturing object goes on the **To Do List** for the following reasons:
 - The object has a nonworking status, or it contains objects which are included in different permission groups. You must change the manufacturing part status back to **Working**, and then update the manufacturing part by using the **Manufacturing Service Manager**.
 - The parent system of the manufacturing object was split in the Molded Forms task, and that split migration produced an error.
 - A compute found a failure within the rule and the manufacturing plate, manufacturing profile, or manufacturing member returned a warning.
 - Some other compute failure, with a specific action per entity, occurred.
- The same behavior occurs when the molded form split is performed by someone who does not have write permissions to the manufacturing part. In this case, a second person with write permission to the manufacturing part must update the manufacturing part by the **Manufacturing Service Manager**.

Manufacturing Service Manager Ribbon

Displays the controls used to update and review manufacturing objects.



Selection of Possible Report Data

Opens a dialog box from which you can set up reports.



Select

Selects objects in the model.

Finish

Starts the **Manufacturing Service Manager**.



Cancel

Clears the current selection.

 **Accept**

Accepts the current selection.

Action

Determines whether the **Manufacturing Service Manager** updates the objects, reviews the objects, or reports the Detailed Review of the objects.

Filter

Specify the type of manufacturing part to use.

- **Parts** - All plate, profile, and member parts.
- **Parts or Assemblies** - All plate, profile, member, and different assembly types. For assemblies, you can select blocks, assembly blocks and assemblies.
- **Plate Parts** - Plate parts.
- **Plate Parts and Connected Profiles** - Includes all profiles connected to the plate parts.
- **Profile Parts** - Profile parts.
- **Member Parts** - Member parts.
- **Systems** - All plate systems, profile systems, and member systems.
- **Systems or Assemblies** - All systems and assemblies.
- **Plate Systems** - All plate systems.
- **Plate Systems and Connected Profiles** - All member systems.
- **Profile Systems** - All profile systems.
- **Member Systems** - All member systems and design member systems.

Connected


Specify which type of connected parts to use.

- **<Ignore>** - Only the selected part.
- **Direct Connected** - Includes objects that are physically connected to the selected part.
- **Indirect Connected** - Includes direct objects and objects that are physically connected to the direct objects.


What do you want to do?

- *Review a manufacturing object* (on page [319](#))
 - *Update a manufacturing object* (on page 319)
 - *Review or update a set of objects selected by a filter* (on page 319)
 - *Detail Review a manufacturing object* (on page 320)
-

Review a manufacturing object


1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing objects to review.
TIP You can select the objects in the model or from the **Workspace Explorer**.
3. Click **Review** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

Update a manufacturing object



1. Click **Select**  on the vertical toolbar.
2. Select the manufacturing objects to review.
TIP You can select the objects in the model, or from the **Workspace Explorer**.
3. Click **Update** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

TIP The software updates the database as each manufacturing object is updated. Clicking **Cancel** stops the update for any objects that have not yet been posted to the database.

Review or update a set of objects selected by a filter

1. Click **More** from the filter selection list.
*The **Select Filter** dialog box displays.*
2. Set up the filter to meet your requirements. For example, you could define a filter to select plate and profile parts in a specific block.
The software finds and highlights the objects that match the filter.
3. Click **Service Manager**  (in the Structural Manufacturing task) or **Tools > Manufacturing Service Manager**.
The software prompts you to use the selected list of objects.
4. Click **Yes**.
5. If required, add additional objects to the list.
6. Click **Review** or **Update** in the **Action** box.
7. Select the type of manufacturing object to review in the **Report Per** box.
8. Click **Finish**.

Detail Review a manufacturing object


1. Click **Select** .
2. Select the manufacturing objects to review in detail.
 **TIP** You can select the objects in the model or from the **Workspace Explorer**.
3. Select **Detailed Review** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

TIPS

- The software reviews the objects and saves a report of each object containing all the other inputs (and their status) to determine the current object status.
- The detailed review report is saved in the Windows temp folder.

Review Dialog Box

Displays the controls used to review the current manufacturing status of manufacturing objects.

 **NOTE** This dialog box opens when you click **Finish** on the **Manufacturing Service Manager Ribbon**.

The manufacturing object status display includes:

- **Up to Date** - Indicates that the object does not require updating.
- **Out of Date** - Indicates that the object requires updating.
- **Can Be Deleted** - Indicates that the object is no longer needed.
- **To-Do-List (Error)** - Indicates that the object is on the **To Do List** because of an error condition.
- **To-Do-List (Out-of-Date)** - Indicates that the object is on the **To Do List** because it requires updating.
- **To-Do-List (Warning)** - Indicates that the object is on the **To Do List** because of a warning condition.
- **Edited** - Indicates that the object has been edited with the part editor. If you modify this part by using the **Manufacturing Plate** or **Manufacturing Profile** commands, or use the **BO Recompute** command, then the status does not display as **Edited**.

Properties

Displays the **Selection of Possible Report Data** dialog box. For more information, see *Selection of Possible Report Data Dialog Box* (on page 321).

Report Per

Displays the current settings for the report data that is set to display. The information that displays depends on what you have set to show or hide on the **Selection of Possible Report Data** dialog box.

Update

Updates the objects. Only the objects shown on the dialog box are updated. This option does not update parts with a status of **Edited**.

Cancel

Closes the dialog box.

File name

Type the path and file name for the report file. You can use the **Browse** button beside the box to browse through the file system.

View

Displays the report file in the default text editor, such as **Notepad**.

Selection of Possible Report Data Dialog Box

Displays the controls used to determine which report data displays on the **Review** dialog box.

Attributes Tab**Option**

Determines the manufacturing object type for which you are setting the review options. Select the box next to the option, and change the setting to **Show** or **Hide** as necessary.

Settings Tab**Review Settings**

Turn on the settings to apply when the **Action** is set to **Review**. The available options include:

- Only include Out of Date manufacturing objects
- Only include manufacturing objects in **To Do List**
- Check detailed part in **To Do List**
- Include light parts
- Refresh after Update
- Highlight Out of Date objects


Update Settings

Turn on the settings to apply when the **Action** is set to **Update**. The available options include:

- Delete objects under light part
- Update objects with detailed part in **To Do List**
- Update manufacturing objects in **To Do List**
- Recompute all manufacturing objects

SECTION 14

New (Block) Assembly


 Creates a new assembly consisting of the parts or other assemblies you select in the graphic view or the **Workspace Explorer**. You can also use the command to create a new assembly block consisting of the parts, assemblies, or blocks you select in the graphic view or the **Workspace Explorer**.



Creating Assemblies


You can think of an assembly as a folder containing other assemblies and parts. To create a new assembly, you must select an existing assembly or block to act as the parent. You can also select parts to assign to the assembly. After you create the new assembly, you can make any changes you require (such as properties, assembly content, or orientation) and save the new assembly to the hierarchy. After you create the assembly, you can change assembly content from the **Workspace Explorer** using drag-and-drop, or you can use the **Cut** and **Paste** commands. **Cut** and **Paste** are available both from the **Edit** menu and the shortcut menu of the **Workspace Explorer**.

★ **IMPORTANT** **Paste** is available for block, assembly, and assembly block objects; **Cut** is only available for assembly objects.

Top-down versus Bottom-up Planning

The default process for creating blocks is based on the concept of top-down planning. By using the **Nested** option in **Split Block** , you create a hierarchy of blocks starting with the top-most block (B0), the entire marine structure, and successively breaking the blocks into smaller volumes that are children to the larger blocks.

Blocks can also be created using a bottom-up planning concept. In this case, all of the blocks are created using the **Flat** option in **Split Block**  so that all of the blocks are at the same level. The block hierarchy is then created by creating assembly blocks using **New (Block) Assembly** . In other words, blocks are initially broken into their smallest components at the bottom of the hierarchy, and combined upwards to create the large blocks.

At some point for both top-down and bottom-up planning, block definition stops and assembly definition begins using **New (Block) Assembly** . The point where this transition occurs is a function of the workflow of the facility and is not determined by the functionality of the software.

New (Block) Assembly Ribbon

Assembly Properties

Activates the **Assembly Properties** dialog box. You can use this dialog box to modify properties such as assembly type or default build equipment, which you cannot set on the ribbon. For more information, see *Assembly Properties Dialog Box* (on page 330).

Selects Sub-assemblies and Parts

Select the assembly or assembly block content. You can select objects in either a graphic view or the **Workspace Explorer**. You can use the **Tools > Hide** command to hide the

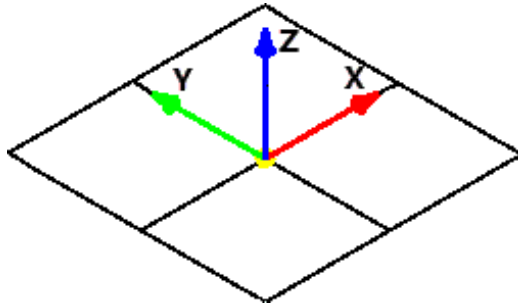
assembly content already selected, thus making it possible to find smaller objects.

Add Secondary Objects

Opens the **Secondary Objects** dialog box. In the dialog box, select the type of secondary objects to include. The secondary objects, such as collars and profiles, are on plates included in the assembly or assembly block.

Manufacturing Orientation

Sets the manufacturing orientation and a viewing direction of the selected assembly or assembly block. When an assembly or assembly block is created, it is given a default manufacturing orientation of marine structure's position as defined by the global coordinate system. The manufacturing orientation plane displays as follows:



The local x-y plane created for manufacturing orientation defines the orientation of the block as it is manufactured on a shop floor. The local z axis is the manufacturing orientation up-vector. By default, it is in the same direction as the global coordinate Z axis.

TIPS

- Manufacturing orientation is also known as assembly orientation.
- Manufacturing orientation is used by Show Assembly View, the **Structural Manufacturing** task, and assembly method drawings in the **Ship Drawings** task.

See the *Manufacturing Orientation Controls* section below.

Finish

Commits the new assembly or assembly block to the database.

Cancel

Cancels the assembly content defined when the **Selects subassemblies and parts** step is active. When the **Select parent assembly or block** step is active, cancels the assembly parent. **Cancel** remains disabled until an assembly or part is selected.

Accept

When the **Selects subassemblies and parts** step is active on the ribbon, clicking **Accept** accepts the assembly content and moves on to the next step. When the **Select parent assembly or block** step is active on the ribbon, clicking **Accept** accepts the selected assembly parent and moves on to the next step. An accept command is needed in order to get a graphical view of the selected parent assembly. **Accept** remains disabled until an assembly or part is selected

Type

Specifies whether the new planning item is an assembly or an assembly block.

Name

Specifies the name of the new assembly or assembly block.

Parent Assembly

Allows you to select the parent assembly or block. Click **More** to choose another assembly or block with the **Select Assembly** dialog box.

Manufacturing Orientation Controls**Orientation Method**





Provides the following options: **Global**, **User Defined**, or **By Object**.

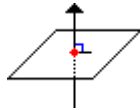
NOTE When the manufacturing orientation is modified, the rotation of the local axes is recomputed according to the following:





1. The software determines which global axis, the X, Y or Z, is closest to the local up-vector, Z.
2. If it is X, then the y-axis is aligned to be as close as possible to Y.
3. If it is Y, then the x-axis is aligned to be as close as possible to X.
4. If it is Z, then the x-axis is aligned to be as close as possible to X.



Global - Orients the assembly according to the global coordinate system (default).

User Defined - Defines the orientation by selecting or creating a plane using one of the following six methods:

-  **Coincident** - Defines a plane coincident to another plane.
-  **Offset from Plane** - Defines a plane at a specified offset distance from another plane.
-  **Angle from Plane** - Defines a plane at a specified angle or slope to another plane. You must define an axis of rotation and the angle or slope.
-  **Plane by Point and Vector** - Defines a plane using a vector normal to the plane being defined and a third point to define the plane position along the vector.



-  **Plane by Three Points** - Defines a plane using three points that you identify in the model.
-  **Plane by Vectors Normal** - Specifies the reference plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.
-  **Remove/Maintain Relationships of the Plane** - Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane**  maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

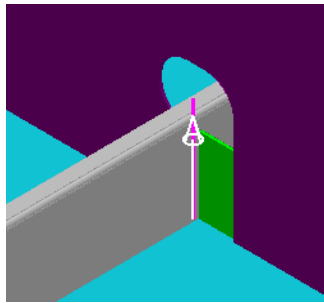
NOTE When you copy a bracket and use **Edit > Paste > Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane**  is not visible. You must first click **Select Bracket Supports**  to allow the command to determine the

applicable options.

By Object - Defines the orientation by specifying a type of object from the **Object Type** list, and then selecting the object in a graphic view or in the **Workspace Explorer**. The supported object types are listed below.

Object Type - Specifies the type of object used to define the orientation.


- **Plate** - Defines the manufacturing orientation of the assembly by selecting the face of a plate. The plate becomes the base plate, the face of the plate becomes the x'-y' plane and the orientation up-vector matches the z' axis. If the selected plate is planar, the selected face is oriented horizontally. If the selected plate is curved, the software sets that plate as the base plate and then defaults to the 3-point method of definition for defining the up-vector. The plate must be a child of the assembly.
- **Pin Jig** - Defines the manufacturing orientation of the assembly by selecting a pin jig. The base plane of the pin jig becomes the x'-y' plane and the orientation up-vector matches the z' axis. The pin jig must be related to one of the children of the assembly.
- **Intersection Seam** - Defines the manufacturing orientation of the assembly by selecting an intersection seam. The orientation up-vector (the z' axis) matches the direction of the seam. This method only allows selection of intersection seams between a plate and a profile. The plate and profile pair that define the seam must be children of the assembly.



- **Seam (By Rule)** - Defines the manufacturing orientation of the assembly by selecting an intersection seam. The orientation up-vector (the z' axis) matches the direction of the seam. This method only allows selection of intersection seams between a plate and a profile that lead to slots on the plate after detailing in the Structural Detailing task. The plate and profile must be children of the assembly and the slot must have a positive opening angle.
- **Coordinate System** - Defines the manufacturing orientation of the assembly by selecting a coordinate system. The orientation up-vector (the z' axis) matches the direction of the coordinate system Z-axis.
- **Assembly** - Defines the manufacturing orientation of the assembly by selecting a sub-assembly (that is, a child assembly in the **Assembly** tab hierarchy of the Workspace Explorer). The orientation up-vector (the z' axis) matches the direction of the up-vector of the sub-assembly.
- **Frame/Axis** - Defines the manufacturing orientation of the assembly by selecting an axis of a coordinate system. The orientation up-vector (the z' axis) matches the direction of the axis.

- **Plane** - Defines the manufacturing orientation of the assembly based on the normal direction of the selected plane. A plane is a grid plane or a planar surface on an object, such as a plate.

Orientation Object - When the **Orientation Method** is set to **By Object**, the list of available objects that can be used to define the orientation is displayed. You can also select one of these objects from the graphic view or the Workspace Explorer.

 **NOTE** **Orientation Objects** is not used by the **Frame/Axis** and **Plane** object types.

Relationship - Specifies the relationship between the selected orientation object and the assembly. If the object is modified, then the manufacturing orientation of the assembly is automatically updated. If the object is deleted, the manufacturing orientation does not change, but the relationship between the object and the assembly is deleted.

- **No Relation** - The software maintains no relationship between the orientation object and the assembly. The orientation object does not display when you select an existing assembly for modification. If you modify the orientation object, the assembly orientation does not change.
- **Relation with Update** - The software maintains the relationship between the orientation object and the assembly. The orientation object displays when you select an existing assembly for modification. If you modify the orientation object, the assembly orientation updates.
- **Relation with No Update** - The software maintains the relationship between the orientation object and the assembly. The orientation object displays when you select an existing assembly for modification. If you modify the orientation object, the assembly orientation does not update. This option is the default value.

Edit View Direction

Allows you to modify the view direction of the assembly when viewing the assembly in manufacturing orientation and assembly method drawings. The view direction symbol is an arrow placed in a quadrant of the manufacturing orientation plane. A default viewing direction is initially provided based on rules. You can change the viewing direction to be used by selecting a new quadrant. The arrow displays in the new quadrant.

X Rotation

Rotates the manufacturing orientation plane with respect to the local x axis. Type a positive value for clockwise rotation about the axis. Type a negative value for counterclockwise rotation about the axis.

Y Rotation

Rotates the manufacturing orientation plane with respect to the local y axis. Type a positive value for clockwise rotation about the axis. Type a negative value for counterclockwise rotation about the axis.

Z Rotation

Rotates the manufacturing orientation plane with respect to the local z axis. Type a positive value for clockwise rotation about the axis. Type a negative value for counterclockwise rotation about the axis.

🔧 Toggle Orientation

Toggles the direction of the local z manufacturing up-vector about the local x-y plane used for definition of the manufacturing orientation plane.




🔍 Fit to Assembly

Fits the graphic view to the assembly.

What do you want to do?

- *Create a new assembly or assembly block* (on page 327)
 - *Set assembly user-defined planar manufacturing orientation* (on page 328)
 - *Set assembly manufacturing orientation by object* (on page 329)
 - *Edit an assembly or assembly block* (on page 330)
-

Create a new assembly or assembly block



1. Click **New (Block) Assembly** .
2. Select content for the new assembly or assembly block from the **Workspace Explorer** or from the model.
3. In the **Type** box, select **Assembly** or **Assembly Block**.
NOTE For **Assembly**, you can also select **Config Project Root** as the parent assembly.
4. In the **Parent Assembly** box, select a parent assembly or block for the new assembly.
5. On the ribbon, click **Add Secondary Objects** .
*The **Secondary Objects** dialog box appears.*
6. Select the types of secondary objects to include in the new assembly and click **Close**.
7. If needed, click **Manufacturing Orientation**  to specify a manufacturing orientation for the assembly.
Set assembly user-defined planar manufacturing orientation (on page 328)
Set assembly manufacturing orientation by object (on page 329)
8. If needed, type a new name for the assembly in the **Name** box.
9. Click **Finish**.

NOTES

- You can also create new assemblies by right-clicking in the **Workspace Explorer** and selecting **New Assembly** from the shortcut menu.
- In the **Workspace Explorer**, you can use drag and drop to change assembly content, or you can use cut and paste.

Set assembly user-defined planar manufacturing orientation

This procedure applies to a new or an existing assembly or assembly block.

1. On the **New (Block) Assembly**  ribbon, click **Manufacturing Orientation** .
2. In the **Orientation Method** box, select **User Defined**.
3. Specify the plane for the manufacturing orientation baseplate using one of the following methods:

Define a coincident plane (on page 39)

Define an offset plane (on page 40)

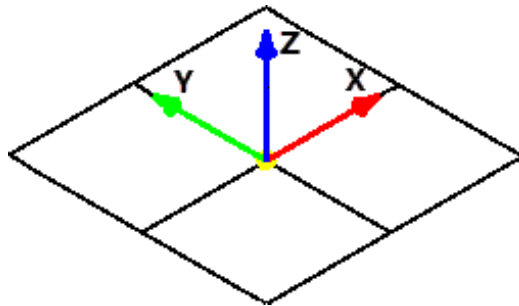
Define plane using angle from plane (on page 41)


Define a plane using a point and a normal vector (on page 41)

Define a plane using three points (on page 42)

Define plane by vectors normal (on page 43)



A graphic of the x-y plane and the z up-vector direction displays in the graphic view.



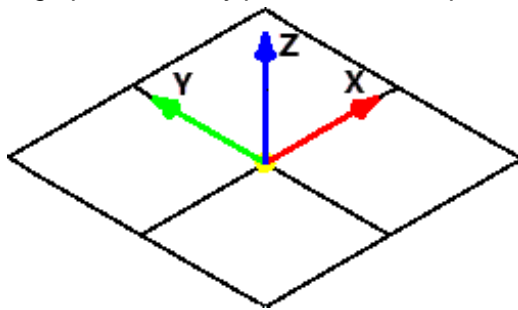
4. If needed, click **Edit manufacturing orientation**  to toggle the direction of the z' up-vector.
5. If needed, type values for **X Rotation**, **Y Rotation**, or **Z Rotation** to rotate the base plate about each coordinate system axis.
6. Click **Finish**.


Set assembly manufacturing orientation by object

This procedure applies to a new or an existing assembly or assembly block.


1. On the **New (Block) Assembly**  ribbon, click **Manufacturing Orientation** .
2. In the **Orientation Method** box, select **By Object**.
3. In the **Object Type** box, select **Plate**, **Pin Jig**, **Intersection Seam**, **Seam (By Rule)**, **Coordinate System**, **Assembly**, **Frame/Axis**, or **Plane**.
4. In a graphic view or the **Workspace Explorer**, select an object appropriate to the selected object type:
 - For **Plate**, select a plate part to use as a baseplate. The normal direction on the baseplate is used as the up-vector.
 - For **Pin Jig**, select a pin jig. The base plane of the pin jig is used as the baseplate. The normal direction on the baseplate is used as the up-vector.
 - For **Intersection Seam**, select an intersection seam between a plate and a profile. The direction of the seam is used as the up-vector.
 - For **Seam (By Rule)**, select an intersection seam between a plate and a profile that lead to slots on the plate after detailing, and the slot has a positive opening angle. The direction of the seam is used as the up-vector.
 - For **Coordinate System**, select a coordinate system. The direction of the Z-axis is used as the up-vector.
 - For **Assembly**, select a sub-assembly of the existing assembly. The orientation of the sub-assembly is used for the assembly.
 - For **Frame/Axis**, select an axis of a coordinate system. The direction of the grid plane or axis is used as the up-vector.
 - For **Plane**, select a plane, such as a grid plane, or a planar surface on a plate. The normal direction on the plane is used as the up-vector.

A graphic of the x-y plane and the z up-vector direction displays in the graphic view.



5. If needed, click **Edit manufacturing orientation**  to toggle the direction of the z' up-vector.
6. If needed, type values for **X Rotation**, **Y Rotation**, or **Z Rotation** to rotate the base plate about each coordinate system axis.
7. Click **Finish**.

Edit an assembly or assembly block

1. Click **Select**  on the vertical toolbar.
2. Select **Assemblies** or **Assembly Blocks** in the **Locate Filter**.
3. Select the assemblies or assembly blocks to modify.
4. Using the ribbon controls, edit the assemblies or assembly blocks as needed.

Assembly Properties Dialog Box

Defines the building method properties for the assembly you are viewing.

General Tab (Assembly Properties Dialog Box) (on page 330)

Relationship Tab (on page 86)

Configuration Tab (on page 86)

Notes Tab (on page 88)

General Tab (Assembly Properties Dialog Box)

Displays and defines the general property names and values of the selected assembly or assembly block. Only information that appears in a field with a white background may be edited. All other fields are read-only.

Category

Select to display a group of assembly properties. Available settings are **Standard, Planning, Weight and CG, Assembly Range, and Position and Orientation**.

Update

Calculates the non-persisted properties, such as length, weight and height.

Standard Category

Name

Name of the assembly. This name is usually generated by the active name rule. You can type a different name for the assembly if you want. Names generated by a rule include a Global Workshare name rule ID, if the name rule ID was defined when the model database was created. For more information, see "Using Global Workshare" in the *Global Workshare Guide*.

Name Rule

Name rule used to generate the name. This field allows you to have optional rules for different situations. The content of the list comes from the "Rule Options" function. Set this option to **User Defined** if you want to specify the name yourself.


Assembly Type

Type of planning assembly, such as block, assembly, or pipe spool. The drop-down list contains all available assembly types as defined in the reference data.

Parent Assembly

Displays the name of the parent assembly, block, or assembly block.

Symmetrically Related Assembly

Displays the name of the symmetrically related assembly if one has been defined using the Mirror Assembly Command . If no relationship is defined, **<None>** is displayed.


Assembly Stage

Construction stage of an assembly from a facility-defined rule set. The drop-down list contains all available stages as defined in the reference data. Examples of stages include part production, assembly of subassemblies, assembly of sections with outfitting, and assembly of sections before painting. If no stage is defined, **<None>** is displayed.


Workcenter

Assigned assembly workcenter from a facility-defined rule set. The drop-down list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Default Build Equipment

Primary equipment used to join the elements of the assembly. The drop-down list contains all available build equipment as defined in the reference data. Examples of build equipment include jig floors and bending equipment. Build equipment can be changed at the assembly element level by clicking **Sequence Form**  on the **Sequence** ribbon. For more information, see Sequence.

Default Weld Equipment

Weld equipment used within the assigned workcenter when welding weld defined using Generate Planning Joints . The drop-down list contains all available weld equipment as defined in the reference data.

Exit Equipment

Crane or transport equipment within the assigned workcenter that moves the finished assembly to its next assembly stage and workcenter. The drop-down list contains all available exit equipment as defined in the reference data.


Building method comments

Allows you to enter comments regarding workcenter or work process assignment or any other issue pertinent to the assembly.


Build And Weld As Sequenced

Specifies if you want to weld objects as they are joined in the assembly sequence. Select **True** to weld each object as it is joined, and select **False** to join all objects together before welding.


Orientation Baseplate

Displays the assigned manufacturing orientation baseplate if one has been defined using the **Manufacturing orientation**  option. If no baseplate is defined, **<None>** is displayed.


Orientation Coordinate System

Allows you to select the global coordinate system used to define the local manufacturing orientation coordinate system in the **Manufacturing orientation**  option. Default value is the main coordinate system.


Orientation X Rotation

The rotation of the assembly around the X-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation**  option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the assembly is no longer oriented by the baseplate.

Orientation Y Rotation

The rotation of the assembly around the Y-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation**  option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the assembly is no longer oriented by the baseplate.


Orientation Z Rotation

The rotation of the assembly around the Z-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation**  option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the assembly is no longer oriented by the baseplate.

Boardside

Board of the assembly. The drop-down list contains all available values as defined in the reference data, typically **Center**, **Starboard Side**, **Port Side**, and **Both Sides**. Default value is **Center**.

Manufacturing Frame System

Allows you to select the coordinate system used to process structural manufacturing rules for all objects in the assembly. The drop-down list contains all available coordinate systems as defined in the reference data. If no coordinate system is defined, **No Framesystem** is displayed and the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation**  option) is used.


Planning Category

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

 **TIP** You can customize slot and collar rules in the Structural Detailing task to use **Build Method** and **Slot Connectivity** properties to determine the slots and collars placed during detailing.

Weight and CG Category

Weight

Displays the weight of all child parts and assemblies of the assembly. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG Coordinate System

Allows you to select the coordinate system for calculation of center of gravity (CoG) coordinates. The drop-down list contains all available coordinate systems as defined in the reference data. Default value is the main coordinate system.

Assembly Range

The range box dimensions of an assembly are based on inputs such as the parts selection criteria, the orientation, the assembly shrinkage, and the assembly margin.

Global - Length

Displays the length of the assembly range box in the global orientation.

Global - Width

Displays the width of the assembly range box in the global orientation.

Global - Height

Displays the height of the assembly range box in the global orientation.

Assembly Orientation - Length

Displays the length of the assembly range box in the assembly orientation.

Assembly Orientation - Width

Displays the width of the assembly range box in the assembly orientation.

Assembly Orientation - Height

Displays the height of the assembly range box in the assembly orientation.

Minimum Bound Box - Length

Displays the length of the assembly range box in the tight box orientation.

Minimum Bound Box - Width

Displays the width of the assembly range box in the tight box orientation.

Minimum Bound Box - Height

Displays the height of the assembly range box in the tight box orientation.

Global - Shrinkage - Length

Displays the length of the assembly range box that includes the assembly shrinkage in the global direction.

Global - Shrinkage - Width

Displays the width of the assembly range box that includes the assembly shrinkage in the global direction.

Global - Shrinkage - Height

Displays the height of the assembly range box that includes the assembly shrinkage in the global direction.

Global - Shrinkage and Margin - Length

Displays the length of the assembly range box that includes the assembly shrinkage and margin in the global direction.

Global - Shrinkage and Margin - Width

Displays the width of the assembly range box that includes the assembly shrinkage and margin in the global direction.

Global - Shrinkage and Margin - Height

Displays the height of the assembly range box that includes the assembly shrinkage and margin in the global direction.

 **NOTE** The length, width, and height values are computed after you click **Update**.

SECTION 15

Submit Batch Job

Tools > Submit Batch Job runs batch processes using the Intergraph Batch Services framework.

This command runs the following types of batch jobs:

Detailing:

- Detail parts
- Undetail parts
- Update reports

Production:

- Generate manufacturing parts
- Generate XML output files
- Manufacturing Service Manager update

Custom batch processes:

- BO recompute
 - Other custom batch processes that you define.
-

What do you want to do?

- *Run a detailing batch process* (on page [336](#))
 - *Run a production batch process* (on page [336](#))
 - *Run a custom batch process* (on page [337](#))
 - *Add a custom batch process* (on page [337](#))
-

Run a detailing batch process

1. Click **Tools > Submit Batch Job**.

*The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

*The **Batch Process Selection** dialog box displays.*

3. Select **Detailing** from the **Batch Process Type** list.

*Smart 3D displays the available detailing batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

*Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

*The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a production batch process

1. Click **Tools > Submit Batch Job**.

*The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

*The **Batch Process Selection** dialog box displays.*

3. Select **Production** from the **Batch Process Type** list.

*Smart 3D displays the available production batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

*Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

*The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a custom batch process

1. Click **Tools > Submit Batch Job**.
*The **Generic Batch Command** dialog box displays.*
2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.
*The **Batch Process Selection** dialog box displays.*
3. Select **Custom Batch PProcesses** from the **Batch Process Type** list.
*Smart 3D displays the available custom batch processes in the **Select Batch Process** list.*
4. Select the process to run from the **Select Batch Process** list, and click **OK**.
*Smart 3D returns to the **Generic Batch Command** dialog box.*
5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.
*The **Schedule [Task] Batch Job** dialog box displays.*
7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Add a custom batch process

1. Add the ProgID and job description of the custom batch process to the <Product Folder>\CommonShip\SOM\Client\Xml\CustomBatchJobDetailsConfiguration.xml file.

For example:

```
<JOB>
  <JobType>CustomBatchProcessD</JobType>
  <JobType_LocaleID>103</JobType_LocaleID>
  <JobDescription>Description of
CustomBatchProcessD</JobDescription>
  <JobDescription_LocaleID>104</JobDescription_LocaleID>
  <JobProgId> ProgID of CustomBatchProcess </JobProgId>
  <IsModelSpecific>False</IsModelSpecific>
</JOB>
```

2. Click **Tools > Submit Batch Job**.
*The **Generic Batch Command** dialog box displays.*
3. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.
*The **Batch Process Selection** dialog box displays.*
4. Select **Custom Batch Process** from the **Batch Process Type** list.

The available custom batch processes display in the **Select Batch Process** list.

5. Select the custom process to run, and click **OK**.

Smart 3D returns to the **Generic Batch Command** dialog box.

6. Specify the **Batch Input Type**, and add inputs as necessary.
7. Click **Schedule**.

The **Schedule [Task] Batch Job** dialog box displays.

8. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Generic Batch Command Dialog Box

Displays the controls used to submit a generic batch command.

Batch Process

Specifies the batch process to run. The list contains recently-selected job types. Click **More** to display the **Batch Process Selection** dialog box, and select a process that is not on the list. For more information, see *Batch Process Selection Dialog Box* (on page 339).

Batch Input Type

Indicates the input type for the batch process.

- **Filter** indicates that you are providing a filter as an input to the batch job.
- **Assembly** indicates that you are providing an assembly as input to the batch job.
- **Object** indicates that you are providing a set of objects as input to the batch job.



Add

Adds inputs to your batch job.

- If **Filter** is selected, this button displays the **Select Filter** dialog box so that you can add filters as inputs to your batch job.
- If **Assembly** is selected, this button displays the **Select Assembly** dialog box so that you can add assemblies as inputs to your batch job.
- If **Object** is selected, this button is not available. To add objects as inputs, select them from the model or the **Workspace Explorer**.



Remove

Removes the selected inputs from the list. The software only removes the inputs that have the associated check box selected.

Share

Saves the inputs for the next batch process. If this check box is cleared, Smart 3D clears the inputs when you select a different batch process.

Schedule as Multiple Jobs

Processes each item in the inputs list as a separate batch process. If this check box is cleared, Smart 3D processes all of the inputs as a single batch process.

Schedule

Displays the **Schedule [Task]** dialog box so that you can schedule the batch process. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Cancel

Exits the command.

Batch Process Selection Dialog Box

Displays the controls used to specify the type of batch process to run.

Batch Process Type

Displays the general batch process types. Select a type to display the available batch process of that type in the **Select Batch Process** list.

Select Batch Process

Displays the available batch process associated with the selected type.

SECTION 16

Manufacturing Export

Controls export of DSTV and XML manufacturing data.

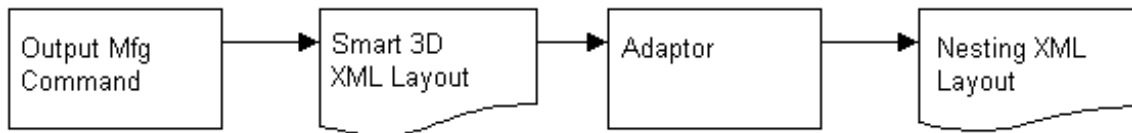
- *Export to Nesting* (on page 340)
- *Managed Export for DSTV* (on page 342)
- *Managed Export for Smart Production* (on page 342)
- *Managed Export* (on page 343)

Export to Nesting

Translates the geometry of selected manufacturing data exchange formats and stores this information in a specified file. The manufacturing data is not saved in the Model database. This command is located on the **Tools > Manufacturing Export** menu.

XML is the default manufacturing data format provided with the software and is used by third-party nesting software. Other formats may be user-defined.

Before finalizing the information, the XML is parsed through an adaptor. The adaptor is a user-defined rule which parses through the default XML schema and allows you to make replacements, modifications, and deletions of entries before finalizing the XML output. This allows customization of the data for different nesting applications.



NOTES

- For built-up profiles, you can generate the profile XML for the web and flange independently of one another.
- You can nest webs or flanges as either plates or flat bars.
- Material type, grade, and thickness are included in the output XML.
- Material type is exported appropriately for the needed nesting software.
- If an object has custom properties, those properties are included in the output XML. For example:

```
<SMS_CUSTOM_PROPERTIES INTERFACE_NAME="IJUAAsemblyChild">
<SMS_CUSTOM_PROPERTY NAME="Build Method" VALUE="5" UNITS_TYPE="long"
PRIMARY_UNITS="undefined" CODELIST="BuildMethodData"
CODELIST_VALUE="VerticalDrop"/>
<SMS_CUSTOM_PROPERTY NAME="Slot Connectivity" VALUE="4" UNITS_TYPE="long"
PRIMARY_UNITS="undefined" CODELIST="SlotConnectivityData"
CODELIST_VALUE="Double"/>
</SMS_CUSTOM_PROPERTIES>
```

- Custom XML is created for the part the XML represents, for the detailed part, for connected detailed objects, for holes, bevels, and margins.
- Free edge treatments on profiles and plates are included in the output XML.

For more information about the XML generated, see the documentation delivered to the *[Product Folder]\Shared Content\XML\StructManufacturing\Doc* folder.

Output Data Ribbon

Displays the controls used to translate the geometry of selected manufacturing data exchange formats and store this information in a specified file.



Select each plate manually

Select manufacturing plates and profiles.



Select all MFG parts in current workspace

Automatically selects all manufacturing parts in the workspace.

Finish

Creates the manufacturing data file.



Cancel

Clears all selected manufacturing parts.



Accept

Accepts all selected manufacturing parts.

Output Format

Select the output format for the manufacturing data file.


Output File



Select the name and location of the manufacturing data file.

Individual Output

Indicates that manufacturing data is output for each part. If this check box is left blank, the manufacturing data is output for the select set. If you choose to output data for each part, the XML file name is identical to the part. If the file already exists, a serial number is attached to the file name to make it unique.

Create a MFG data file

1. Click **Tools > Struct Manufacturing > Export to Nesting**.
2. On the **Output Data** ribbon, click **Select each plate manually** .
3. Select manufacturing plate or profile parts.

 **TIP** You can click **Select all MFG parts in current workspace**  if you want to select all the manufacturing parts.
4. Select a value from the **Output Format** list.
5. Select a value from the **Comparison Rule** list.

6. Select a location and name for the output file.
7. Click **Finish**.

Structural Manufacturing XML Schema

The Structural Manufacturing XML Schema is documented in the **Symbols** share in the *[Reference Data Folder]\Data\Symbols\XML\StructManufacturing\Doc* folder.

The schema is delivered to the **Symbols** share in the *[Reference Data Folder]\Data\Symbols\XML\StructManufacturing* folder.

Profile XML Schema

The Profile XML Schema is documented in the **Symbols** share in the *[Reference Data Folder]\Data\Symbols\XML\StructManufacturing\Doc* folder.

The schema is delivered to the **Symbols** share in the *[Reference Data Folder]\Data\Symbols\XML\StructManufacturing* folder.

See Also

Export to Nesting (on page 340)

Create a MFG Data File (on page 341)

Managed Export for Smart Production

Exports both the DSTV and XML data. This export uses the SmartProductionConfig.xml default configuration file. This command is located on the **Tools > Manufacturing Export** menu.

Smart 3D delivers example configuration files for managed exports to the *[Reference Data Folder]\SharedContent\StructManufacturing\ManagedExport* folder.

This command uses the **Managed Export** dialog box, automatically specifying the SmartProductionConfig.xml file. For more information, see *Managed Export Dialog Box* (on page 343).

Managed Export for DSTV

Exports the DSTV data for the selected assembly, and manages the .nc1 files that the software generates. This export uses the DSTVConfig.xml default configuration file. This command is located on the **Tools > Manufacturing Export** menu.

Smart 3D delivers example configuration files for managed exports to the *[Reference Data Folder]\SharedContent\StructManufacturing\ManagedExport* folder.

This command uses the **Managed Export** dialog box, automatically specifying the DSTVConfig.xml file. For more information, see *Managed Export Dialog Box* (on page 343).

Managed Export Dialog Box

Controls parameters for exporting DSTV and manufacturing XML data for assemblies.

Configuration File

Displays the path and file name for the current configuration file. This configuration file drives the managed export process. This is a required field for all types of managed exports. For managed exports for DSTV, Smart Production, and SMS_SCHEMA, the software provides a default configuration file. For neutral managed exports, you must specify the configuration file. Click **Browse** next to the **Configuration File** box to navigate to the configuration file.

Export Type

Displays the type of data that you are exporting. The list contains options as defined in the configuration file. You can use this option to organize your export by type of data (plates, profiles, panels, templates, and so on), by location (host location or satellite location), or any other category that you require.

Entities

Displays information about previously exported assemblies.

Process – Indicates whether the block is exported. Select the option to add the block to the export. Clear the option to avoid exporting the block.

Update - Updates the export.

Extraction - Creates output.

Reference Assembly - Displays the name of the parent block or assembly.

Assembly - Displays the name of the assembly that the software is processing.

Versioning - Displays the version of the export for the block.

Status - Displays the status of the output.

Output Location – Displays the path and filename of the export .xml file.

Export

Starts the export process based on the selected configuration file.

Managed Export

Creates a neutral export. This export does not have an associated configuration file. Instead, it uses the configuration file that you select. This export is intended to be used with custom configuration files. This command is located on the **Tools > Manufacturing Export** menu.

Smart 3D delivers example configuration files for managed exports to the *[Reference Data Folder]\SharedContent\StructManufacturing\ManagedExport* folder. You can create your own export file, but it must follow the format provided in the example files.

This command uses the **Managed Export** dialog box, but does not automatically specify a .xml file. For more information, see *Managed Export Dialog Box* (on page 343).

SECTION 17

Manufacturing Services

This command helps you create customized reports for specific manufacturing objects. These reports can have different file formats (such as *.txt or *.xml). The **Custom Processing** command gets the object types that can be reported and the processes that create the reports from selected manufacturing objects from a configuration file named **MfgCustomProcessOptions.xml** which is located in the [Symbol Server]\Xml\StructManufacturing\CustomProcess folder.

This command is located on the **Tools** menu.

Configuration File

The configuration file is centralized at the symbol share location. On the symbol share, the **MfgCustomProcessOptions.xml** file is stored in the [Symbol Server]\Xml\StructManufacturing\CustomProcess folder.

You can customize this XML file to add various reports, by using the following:

Name

The name of the filter set defined by the **Identifier**.

Identifier

A filter to the command. The properties set here act as the filter to the custom program.

Process name

A user-defined name of the custom program that creates the report.

Process id

The Prog ID of the custom program to execute.

FileType

The output type for the file. The custom command must save this information.

<MfgProcessSet>

The XML node used to control the different object entries.

<Object>

XML nodes for one object report type entry.

<process/>

XML nodes for one custom report entry.

XML Schema

The XML schema has the following format:

```
<?xml version="1.0" ?>
<MfgProcessSet>
  <object      name=" " Identifier=" ">
    <process    name=" " processid=" " FileType=" " />
    <process    name=" " processid=" " FileType=" " />
  </object>
</MfgProcessSet>
```

Report Program

The following API interface generates a report.

```
Private Sub IJDCustomReport_Generate(
    ByVal pElements As GSCADStructMfgGlobals.IJElements,
    strFileName As String,
    eCustomReportStatus As GSCADStructMfgGlobals.CustomReportStatus)
```

Examples are delivered to the *[Product Folder]\StructManufacturing\Data\Rules\CustomReport* folder.

Custom Processing Ribbon

Displays the controls used to run a custom processing executable.



Select

Select input objects based on the selection in the **Object Type** field.

Finish

Run the custom processing command.



Cancel

Clears the selection.



Accept

Accepts the selection.

Object Type

Specifies the filter to use.

Process

Specifies the name of the process to use.

Output



Specifies the output type.



File Open

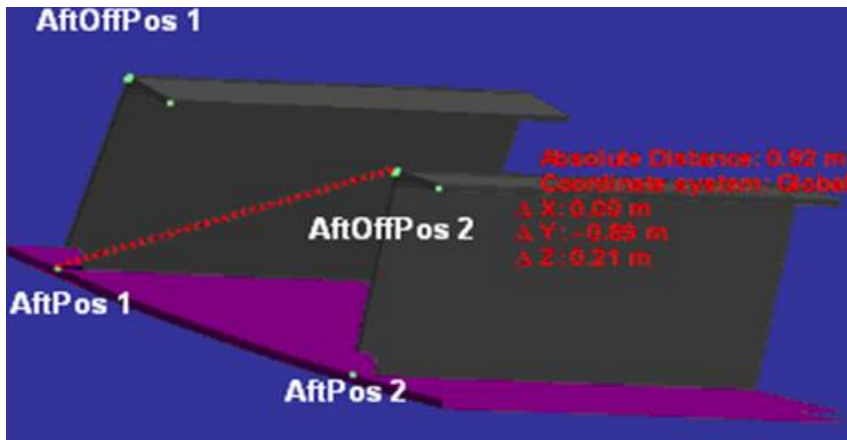
Opens a file into which the custom command writes the output.

Run a custom processing command

1. Click **Tools > Custom Processing**.
2. On the **Custom Processing** ribbon, select the type of object to process from the **Object Type** list.
3. Select the process to run from the **Process** list.
4. Select the objects on which to run the process from the graphics view or the **Workspace Explorer**.
5. Click  **File Open** and specify a file for the output.
6. Click  **Accept**.
7. Click **Finish**.

Stiffeners Diagonal Measurement

Provides a tool to extract distance information between two stiffeners or a pin jig for accuracy verification. This tool also provides information on mounting angles. This command is located on the **Tools** menu.



Stiffeners Diagonal Measurement Ribbon

Displays the controls used to extract diagonal lengths between stiffeners, or for pin jigs.

Selection By

Indicates whether the diagonals are between stiffeners, or for pin jigs.



First Stiffener

Select the first stiffener for the diagonal. This option is available only when the **Selection By** list is set to **Stiffeners**.



Second Stiffener

Select the second stiffener for the diagonal. This option is available only when the **Selection By** list is set to **Stiffeners**.

Pin Jig

Select the pin jig for the diagonal. This is optional when the **Selection By** list is set to **Stiffeners**.

View

Displays the **Diagonal Lengths and Mounting Angle** dialog box.

Diagonal Lengths and Mounting Angle Dialog Box

Displays the controls used to generate reports on distances and mounting angles.

Select Seam

Select a seam from the list of outer boundaries of the pin jig. This option is only available if a pin jig is selected.

Distance

Specify a distance to identify which stiffeners are bounded to the selected seam. This option is only available if a pin jig is selected.

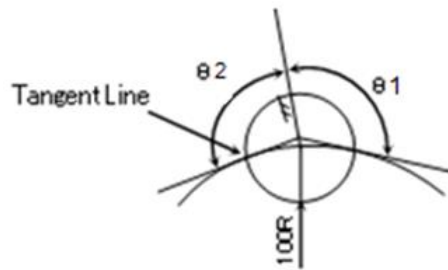
Overlap Factor

Specifies the minimum overlap factor for the stiffeners on the pin jig remarking surface.

Angle Report Type

Specifies the method for reporting mounting angles. The available options are:

- **Based on Surface** - Report mounting angle with respect to the stiffening-surface normal or the base-plane normal.
- **Based on Tangent** - Report mounting angle by constructing tangents at the circle intersection points with the stiffening or remarking surface.



Convex Radius

Specifies the radius used for mounting angle computation when the curvature of the surface is convex. This option is available only if the **Angle Report Type** is set to **Based on Tangent**.

Concave Radius

Specifies the radius used for mounting angle computation when the curvature of the surface is concave. This option is available only if the **Angle Report Type** is set to **Based on Tangent**.

Select First Stiffener

Select the first stiffener from the list of stiffeners.

Select Second Stiffener

Select the second stiffener from the list of stiffeners.

First Stiffener**Points to Process**

Select either **Length Points** or **Vertex Points** from the list. Length points are points defined by the symbol file. The symbol file is then used to prepare the end cut. Vertex points are points defined by the base or offset edge of the stiffener.

Stiffener Side

Select the point from which to measure. If the **Points to Process** option is set to **Vertex Points**, then the available options include **Base - Web Left**, **Base - Web Right**, **Base - All**, **Offset Web Left**, **Offset - Web Right**, and **Offset - All**. If the **Points to Process** option is set to **Length Points**, then the options include **Base - Web Cuts**, **Base - Flange Cuts**, **Base - All**, **Offset - Web Cuts**, **Offset - Flange Cuts**, and **Offset - All**.

Points

Select the point on the first stiffener from which to measure. The point names represent the closest two edges of the stiffener. For example, **WL_B_Point** indicates the point on the left side of the web and bottom of the stiffener. The point highlights in the model.

**SecondStiffener****Points to Process**

Select either **Length Points** or **Vertex Points** from the list. Length points are points defined by the symbol file. The symbol file is then used to prepare the end cut. Vertex points are points defined by the base or offset edge of the stiffener.

Stiffener Side

Select the point from which to measure. If the **Points to Process** option is set to **Vertex Points**, then the available options include **Base - Web Left**, **Base - Web Right**, **Base - All**, **Offset Web Left**, **Offset - Web Right**, and **Offset - All**. If the **Points to Process** option is set to **Length Points**, then the options include **Base - Web Cuts**, **Base - Flange Cuts**, **Base - All**, **Offset - Web Cuts**, **Offset - Flange Cuts**, and **Offset - All**.

Points

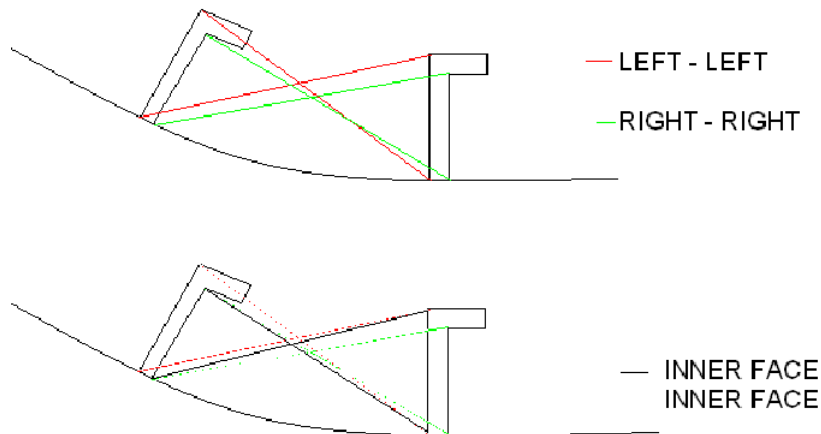
Select the point on the second stiffener from which to measure. The point names represent the closest two edges of the stiffener. For example, **WL_B_Point** indicates the point on the left side of the web and bottom of the stiffener. The point highlights in the model.

Parameter - Value table

Displays the distances and mounting angles based on the points that you have selected.

View/Report Options

Select the type of report to generate. Available options include **Web left-Web left**, **Web right-Web right**, and **Inner face-Inner face**.

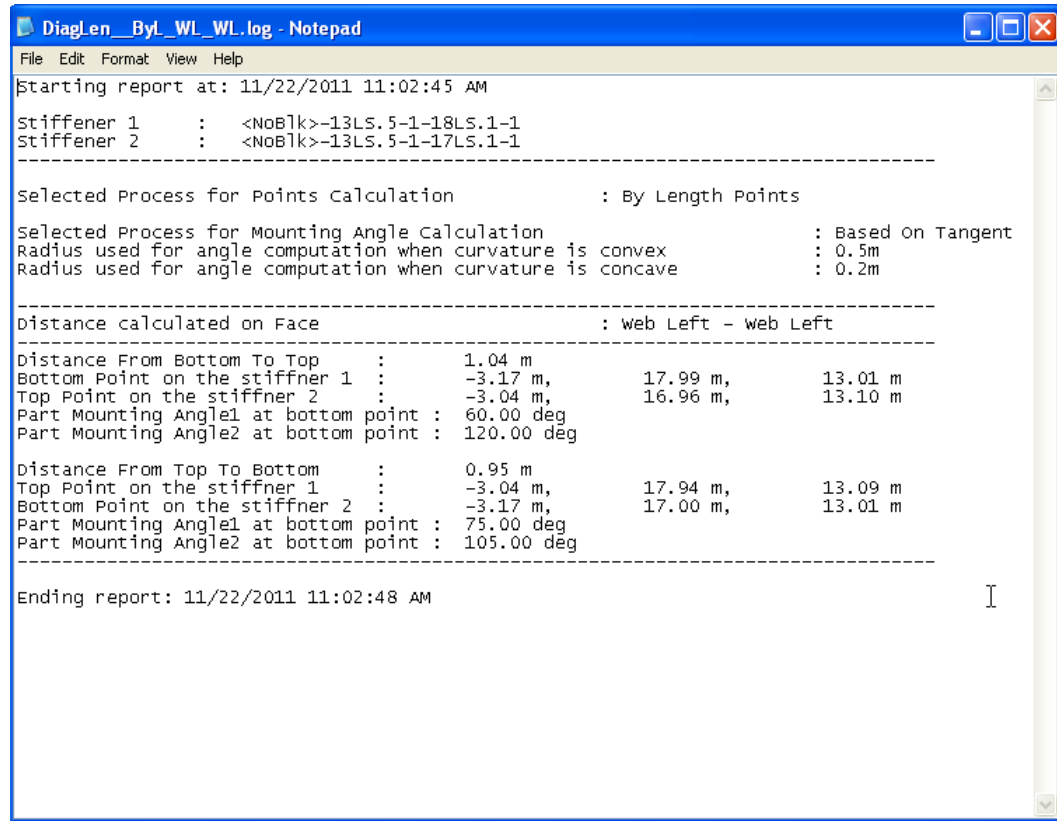


Fit View

Fits the view to the selected objects.

Generate Report

Creates a report on distances and mounting angles. Additionally, you can view the generated report.



```

DiagLen_ByL_WL_WL.log - Notepad
File Edit Format View Help
Starting report at: 11/22/2011 11:02:45 AM
Stiffener 1 : <NoBlk>-13LS.5-1-18LS.1-1
Stiffener 2 : <NoBlk>-13LS.5-1-17LS.1-1
-----
Selected Process for Points Calculation : By Length Points
Selected Process for Mounting Angle Calculation : Based On Tangent
Radius used for angle computation when curvature is convex : 0.5m
Radius used for angle computation when curvature is concave : 0.2m
-----
Distance calculated on Face : Web Left - Web Left
-----
Distance From Bottom To Top : 1.04 m
Bottom Point on the stiffner 1 : -3.17 m, 17.99 m, 13.01 m
Top Point on the stiffner 2 : -3.04 m, 16.96 m, 13.10 m
Part Mounting Angle1 at bottom point : 60.00 deg
Part Mounting Angle2 at bottom point : 120.00 deg
-----
Distance From Top To Bottom : 0.95 m
Top Point on the stiffner 1 : -3.04 m, 17.94 m, 13.09 m
Bottom Point on the stiffner 2 : -3.17 m, 17.00 m, 13.01 m
Part Mounting Angle1 at bottom point : 75.00 deg
Part Mounting Angle2 at bottom point : 105.00 deg
-----
Ending report: 11/22/2011 11:02:48 AM

```



Select Connected Object

Provides a tool to check which objects are connected to a selected object based on physical connections between the parts. You can determine if a given manufacturing plate or profile is out of date. The information shown by this command is the same list as that shown on the **Status** tab when you modify a single manufacturing plate or profile. This command is located on the **Tools > Struct Manufacturing** menu. You must select the object, and then start the command.

You can also use **Ctrl + Shift + M** to start this command from the Structural Manufacturing task.

NOTE In version 7.4, you must start this command as a custom command using the ProgID **GSCADStrMfgStatusCmd.ConnectionCmd**.

Display connected objects

1. Select the object for which to display connected objects.
 **TIP** You can select a detailed plate or profile or a manufacturing plate or profile.
2. Click **Tools > StructManufacturing > Select Connect Object**.
 **TIPS**
 - You can also press **Ctrl + Shift + M** to start the command.
 - In version 7.4, you must start this command as a custom command using the ProgID **GSCADStrMfgStatusCmd.ConnectionCmd**.

*The **Dependent and Connected Objects** dialog box displays.*
3. Select the objects to highlight from the grid.
4. If necessary, click **Show Properties** to display the **Properties** dialog box for the selected object.

Dependent and Connected Objects Dialog Box

Dependent Objects

Displays a list of the dependent objects.

Connected Objects

Displays a list of the objects connected to the dependent object, if available. The connected objects have physical connections because they have profiles or plates as connected objects.

 **NOTE** Click a cell to highlight that object in the model.

Show Properties


Displays the **Properties** dialog box for the selected object.

Highlight Physical Connections

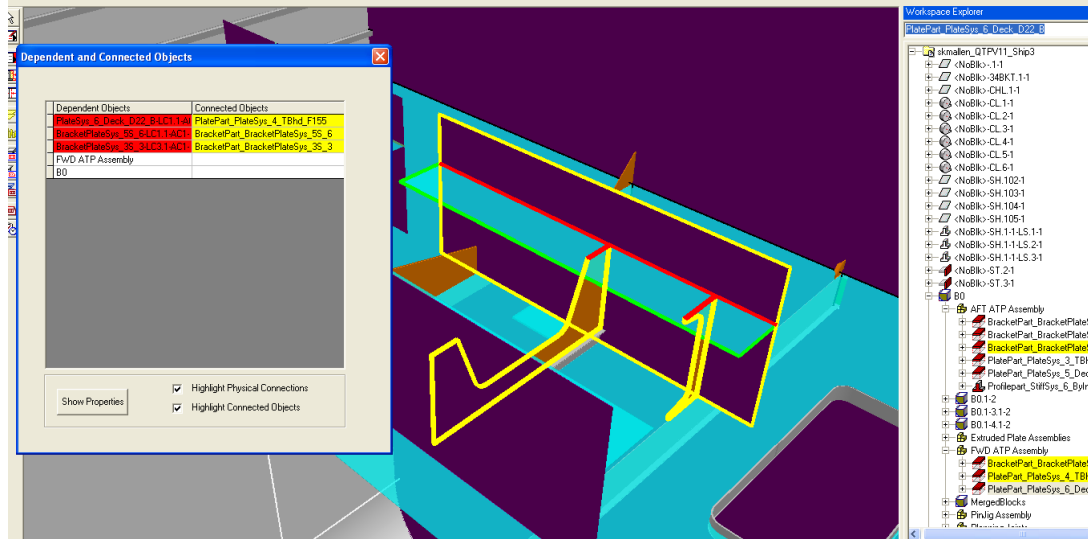
Highlights the physical connections associated with the selected objects in the model.

Highlight Connected Objects

Highlights the objects connected to the selected objects in the model.

 **NOTE** The highlighted objects in the model use the following colors:

- Green - Selected object
- Red - Dependent objects



Provides a tool to view, edit, and overwrite generated macros placed on profile and member parts. The software displays profile and member parts and the macros generated for the parts in a grid. You can edit macro attributes in the property grid, or apply a different macro to a part. You can also reset the definition and attributes of a macro to the rule-selected values. This command is located on the **Tools** menu under **Manufacturing Services**.

Select **Manufacturing Services > Macro Command** from the **Tools** menu to display the **Macro Editor**.

Drag a column header and drop it here to group by that column

Groups the macro commands by the column header you select. Drag a column header to the space above the **List of Stiffeners/Members** to group the macros by the selected column. Click the column in the **Grouped by** space to change the sort order for that group. Drag the column back to the **List of Stiffeners/Members** area to ungroup the macros. Alternatively, click the 'x' in the column group box to ungroup the macros.



Column Chooser

Displays a list of columns that you can show or hide under **List of Stiffeners/Members**. The available columns depend on whether the macro column grid is populated. By default, the **Part Name**, **Cross Section**, **Cross Section Name**, and **Standard/Version** columns are available. Additional columns display when you select a macro from the column grid. Clear a column check box to hide that column.

List of Stiffeners/Members

NOTES

- Click a column header to change the sort order for that column.

- Click the filter  in a column header to define arguments that narrow the list of macros in that group.
- If you do not see one of the columns listed below, click **Column Chooser**  and select the check box for the hidden column.

Part Name

Displays a list of part names associated with the selected assembly, profile, or member.

Cross Section

Displays the cross-section associated with the part name in the current row.

Cross Section Name

Displays the cross-section name associated with the part name in the current row. Sections are defined in the reference data. For more information about reference data, see the *Structure Reference Data Guide*.

Standard/Version

Displays the standard and the version of the standard that contains the macro definition.

NOTES

- The commands listed below are unavailable if the macro command grid is empty.
- The software does not populate the macro command grid until you select an assembly, a profile, or a member either from the **Workspace Explorer** hierarchy or from the Smart 3D model. For more information, see *Selecting Objects* (on page 19).
- When the software populates the command grid, the column headers also display in the **Column Chooser**.

Features**End Cut at Start**

Displays the macro for the corresponding part before end-cuts remove material from the profile solid.

End Cut at End

Displays the macro for the corresponding part after end-cuts remove material from the profile solid.

Macro command grid

Displays the stiffeners and members for the loaded assembly, profile, or member. If a macro attribute is modified, it displays in the command grid with a yellow background. If a macro is overwritten (that is, the macro definition is completely changed), the macro has a red background.

NOTE When you select a macro in the command grid, the **Macro Editor** displays the following commands on the right side of the **Macro Editor**.

Categorize 

Sorts the macro properties by category.

Alphabetical 

Sorts the macro properties alphabetically.

Search box

Allows you to search for a macro property.

Macro Properties

Displays properties for the selected macro. You can select a different macro type from the **Name** list, however the other properties are read-only.

Name - Select a macro type from the list to change the macro for the selected member.

Macro Output Properties


Displays the values for each of the macro output properties. You cannot edit these properties.

Parameters

Displays parameters for the selected macro property. Click a parameter field and type a new value to modify the parameter.

Display Macro Image

Displays a preview of the macro image so that you can review changes before accepting them. The macro image automatically updates when you change the macro name and then click **Save**.


 **TIP** To display the macro image in a separate, sizable window, drag the macro image to the Smart 3D window. The software clears the **Display Macro Image** check box, and the check box is unavailable until you close the separate macro image window.

Reset

Returns all of the current macro settings to the rule-generated version.

Save

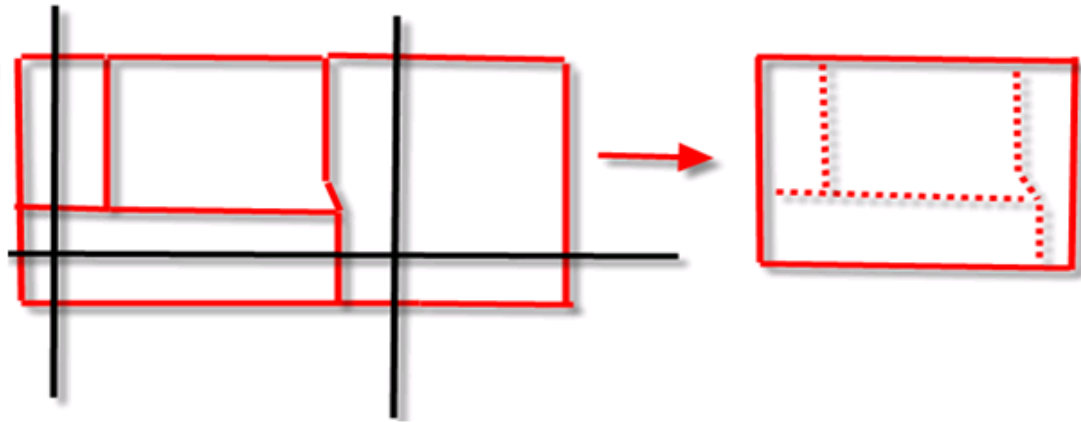
Saves changes to the selected macro.

 **NOTE** If you select another macro before saving changes to the current macro, the software prompts you to save or discard the current changes.

Copy Structure

Provides a tool to generate new plate geometry by copying from an original plate with advanced plate system marks representing the connected structure. This allows you to produce the manufacturing data bypassing any problems from the original model data set. This command also provides a tool to generate a surface from plates that can be taken to GAR and repaired before it is used for manufacturing data. This command is located on the **Tools > StructManufacturing** menu.

NOTE In version 7.4, you must start this command as a custom command using the ProgID **StrMfgCopyStructCmd.CopyStructure**.



Copy Structure Ribbon

Displays the controls used to generate a plate or surface by copying existing geometry.

Properties

Activates the **Copy Structure Properties** dialog box, which you use to view and modify the properties of the copy that you are about to place in the model. For more information, see *Copy Structure Properties Dialog Box* (on page 357). This option is not available if **Action Type** is set to **Generate Surface**.

Plate Parts

Select the detailed plate parts to copy. You can select the parts in a graphic view or in the **Workspace Explorer**.



Pick Boundaries

Define boundaries by selecting them graphically either in a graphics view or in the **Workspace Explorer**. Seams and frames are allowed as boundaries.

Finish

Displays a dialog box asking if you want to accept or reject the changes. If you click **Accept**, the software displays a preview of the template set. Creates the template using the parameters that you have defined.

Solve Ambiguity

The command switches to **Solve Ambiguity**  automatically if an ambiguous solution exists after clicking **Accept**  or **Finish**. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 45).

Cancel

Clears the selection.

Accept

Accepts the selection.

Action Type

Specifies the type of copy to make. The following options are available:

- **Generate Plate** - Creates a new plate part from the selected plates and boundaries.
- **Generate Surface** - Creates a new surface from the selected plates and boundaries based on the selected **Plate Side** and **Surface Type**. The new surface is exported to a SAT file.
- **Generate Unfold Data** - Creates unfolded data from the selected plate system and boundaries. The selected surface is exported to a SAT file.

Up Side

Select the side on which to create the plate. This option is only available if the **Action Type** is set to **Generate Surface**. The following options are available:

- **Molded Side**
- **Anti Molded Side**
- **Base Side**
- **Offset Side**

Surface Operation

Select the type of surface to place. This option is only available if the **Action Type** is set to **Generate Surface**. The following options are available:

- **Detailing Neutral Surface**
- **Detailing Part Surface**
- **True Molded Surface**
- **True Part Surface**
- **Detailing Molded Surface**

**SAT File**

Specify the SAT file into which to write the surface. This option is only available if the **Action Type** is set to **Generate Surface** or **Generate Unfold Data**.

Frame System

Contains a list of frame system names that are in the range of plate part. The frame marks are generated on the unfolded surface, based on the frame system selection. This option is only available if the **Action Type** is set to **Generate Unfold Data**.

Copy Structure Properties Dialog Box

Specifies the properties for the copy structure that you are editing.

See Also

General Tab (Copy Structure Properties Dialog Box) (on page 357)

Marks Tab (Copy Structure Properties Dialog Box) (on page 357)

General Tab (Copy Structure Properties Dialog Box)

Standard

Name

Specifies the name of the copy structure. If you change the name, then the name rule changes automatically to **User Defined**. The default value is **Copy of <Old plate system name(parent of the plate picked)>**. If you pick multiple plates, the names are separated by an underscore (_).

Generic System

Specifies the name of the generic system object for the new plate.

Parent System

Select the parent system for the new generic system (if given) or plate (if generic system is not given).

Parent Assembly

Select the assembly for the new plate parts.

Actual Thickness

Specify the actual thickness value. The default value is the thickness of the selected plate.

Split with Inner Seams

If set to **No**, the software does not split the new plate along the inner seams. If set to **Yes**, the software splits the plate.

See Also

Copy Structure Properties Dialog Box (on page 357)

Marks Tab (Copy Structure Properties Dialog Box)

Displays a list of custom attributes. Typically, the attributes can be set to **Apply** or **Ignore**. These values are codelisted, and they can have a rule ProglID associated to them as the long description. These attributes are various advanced plate system marks for corresponding connections.

Reference Curve Marks

Displays reference curves on the selected plate parts.

Plate Location Marks

Displays connected plates on the selected plates.

Profile Location Marks

Displays profiles stiffening the selected plates.

Collar Marks

Displays connected collar plates.

Bracket Marks

Displays connected brackets.

Lap Connection Marks

Displays connected lapped plates.

End Connection Marks

Displays profiles connected on their base or offset surface.

Inner Seam Marks

Displays seams between the selected plate parts.

Outer Seam Marks

Displays seams at the resulting outer contour of selected plates.

See Also

Copy Structure Properties Dialog Box (on page 357)

SECTION 18

Check Manufacturability

Tools > Check Manufacturability (**Tools > Check Hole** in the **Hole Management** task) analyzes objects in the current workspace and reports the objects that will be difficult or impossible to manufacture according to standards defined in the reference data.

Check Manufacturability Ribbon

Provides the following options to check that objects in the model can be manufactured and to generate production information.

Manufacturability Checking Settings

Shows the rules for the checking process. You can select any or all of the rules. For more information, see *Manufacturability Checking Settings Dialog Box* (on page 361).

Check Manufacturability

Starts the checking process. If inconsistencies are found, the message **Manufacturability errors/warnings have been encountered** displays.

Show Manufacturability Inconsistencies

Displays the objects with manufacturing issues. The list includes the issue severity, the assembly name, a description of the assembly, and the name of the rule which detected the manufacturing issue. For more information, see *Show Manufacturability Inconsistencies Dialog Box* (on page 362).

Close

Exits the command.

Submit Job

Displays a message box asking if you want to execute the repair rules subsequently. If you click **Yes**, the **Schedule Planning Check Manufacturability** dialog box displays so that you can schedule the execution of the repair rules at some other time. For more information, see *Schedule [Task] Dialog Box* in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.




NOTES

- Learn more about SmartPlant Batch Services in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.
- The **Submit Job** button is enabled only when Batch Services is installed and started on your computer.
- The data on objects that cannot be manufactured is stored in the session file, not in the model. Problem objects that apply to the Workspace are retrieved when you select the **Check Manufacturability** command.


What do you want to do?

- *Check objects for manufacturability* (on page 360)
 - *Schedule check manufacturability using Batch Services* (on page 360)
 - *View the Manufacturability Checking Settings Dialog Box* (on page 361)
 - *View the Show Manufacturability Inconsistencies Dialog Box* (on page 362)
-

Check objects for manufacturability

1. Select objects for checking in the model or in the **Workspace Explorer**.
2. Click **Tools > Check Manufacturability** (**Tools > Check Holes** in Hole Management).
3. On the ribbon, click **Manufacturability Checking Settings** .
4. Specify the rules for the checking process.
5. On the ribbon, click **Check Manufacturability**  to start the process.
6. On the ribbon, click **Show Manufacturability Inconsistencies** .
A list view displays the objects that were found to have manufacturing problems.
7. Select one row of the list at a time to view the inconsistency and solution for an object.
8. Repair the object manually, or click **Repair** if an **Action Tool** is available.

Schedule check manufacturability using Batch Services

1. Select objects for checking in the model or in the **Workspace Explorer**.
2. Click **Tools > Check Manufacturability**.
3. On the ribbon, click **Manufacturability Checking Settings** .
4. Specify the rules for the checking process.
5. Click **Submit Job**.
6. Click **Yes** to schedule the job for another time.
7. Complete the **Schedule Check Manufacturability** dialog box to configure batch processing, and then click **OK**.

The software displays the message, "This job has been scheduled to run on the chosen server."

8. Click **OK** to dismiss the message.

NOTES

- For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.
- After the job is successfully submitted, you can see the PlanningCheckMfcty_BatchJob.xml file in the system temp folder (%temp%). This file contains the required information to run the batch job.
- If any inconsistencies are observed during the execution of the check manufacturability rules that you selected, information about the name of the object, manufacturability rule ProgID, and repair rule ProgID are written to the PlanningCheckMfcty_InConsistencies.log file in the system temp folder (%temp%).

Manufacturability Checking Settings Dialog Box

Sets options for the **Check Manufacturability** process.

Task List


Displays the list of tasks that contains the **Check Manufacturability** rules defined in the catalog. These tasks allow you to filter the check rules based on the selected task in the **Task List**.

Rule

Select which rules to process check manufacturability against. There are different rules based on the task selected in the **Task List** option. Contact Intergraph Support <http://support.intergraph.com> for help with customizing the XML data file.

- The **Hole Management** manufacturability checking process includes checking the distance between hole traces; the distance between the hole trace and plate edges (outside), brackets and coamings; and the distance between the hole trace and seamlines, profiles, and openings. Because hole traces can affect the stress of the deck or bulkhead on which they are placed, it is important to check holes before cutting them. For more information about hole management checks, see Check Hole Rules in the *Hole Management Reference Data Guide*.
- The **Piping** manufacturability checking process includes software simulators for cutting, bending, and coating pipe. An XML data file controls the piping simulators. You can customize this file to reflect the requirements, standards, and equipment of your facility. For more information about piping checks, see Piping Check Manufacturability in the *Piping Reference Data Guide*.
- The **Planning** manufacturability checking process includes software simulators for planning joints, weld bevels, and slot types on structure. You can customize the planning simulators rules to reflect the requirements, standards, and equipment of your facility. For more information about planning checks, see Check Manufacturability Rules in *Planning Reference Data*.
- The **Structural Detailing** checking process includes tee weld chamfer checks based on changes to plate thickness, slot opening angles, and end cut type. For more information, see Check Manufacturability Rules in *Structural Detailing Reference Data*.
- The **Structural Manufacturing** checking process includes margin check on the same port, manufacturing plat and profile check, and shrinkage check.

ProgID

Programming ID of the rule that found the inconsistency. Matches the **Rule ProgID** of the rule on the *Show Manufacturability Inconsistencies Dialog Box* (on page 362) .

Select All

Specifies all available rules.

Clear All

Clears all available rules. You can select rules individually in the list view by clicking the boxes beside the rule names.

Stop checking if a manufacturability error is found

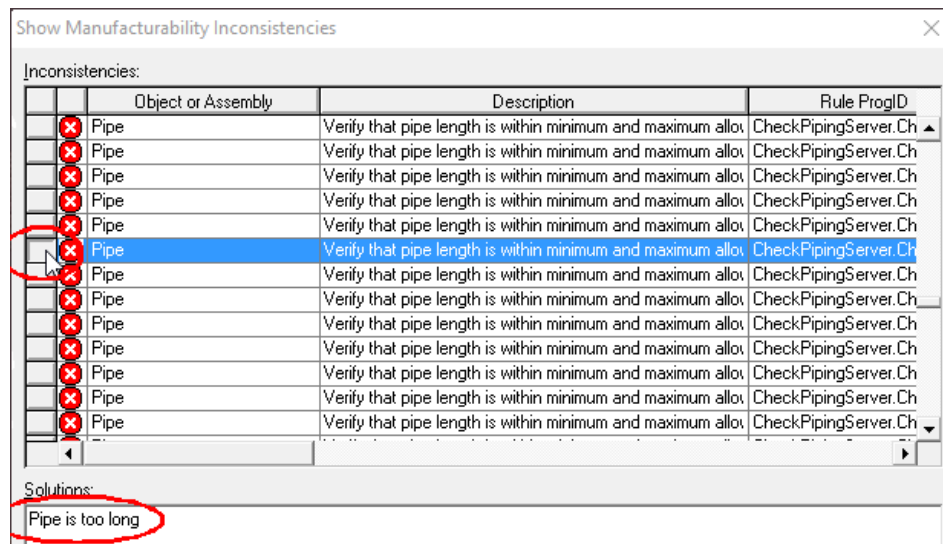
Halts rule processing upon error detection.

Show Manufacturability Inconsistencies Dialog Box

Displays and repairs inconsistencies found by the rules of the **Check Manufacturability** process.

Inconsistencies

A list view of all inconsistencies. Click the button in the far left column to see the possible solution. Solution descriptions are listed below.



 **Warning**

A minor manufacturability problem has been found. The problem does not have to be repaired.

 **Error**

A major manufacturability error has been found. The error must be repaired.

Object or Assembly

Name of the object with an inconsistency.

Description

Description from the rule of the inconsistency

Rule ProgID

Programming ID of the rule that found the inconsistency. Matches the **ProgID** of the rule on the **Manufacturability Checking Settings Dialog Box** (on page 361) .

Action

The type of action that is available in the software to repair the inconsistency. Blank if no software action is available and the repair is done manually.

- **COM Repair Object** - Software code that can be run.
- **SQL Script** - SQL Script that can be run.

Action Tool

ProgID of the COM Repair Object or name of the SQL Script. Blank if **Action** is blank.

Solutions

Describes the details of an inconsistency and the repair that you need to perform for one selected object from the list.


Repair

Runs the **Action Tool** to perform a repair, if one is available for the rule.

Fit

Fits one or more selected objects from the list view in the active graphic view.

Clear

Clears all inconsistencies and closes the dialog box. You must click **Check Manufacturability**  again to display the remaining inconsistencies.

Piping Inconsistencies Solution Messages**Pipe and Spool Length Messages****Pipe is too long**

Pipe exceeds standard ordering length for pipe of this size, as defined in the pipe catalog. For bent pipes the length check includes any extra pipe that had to be added at the ends or between bends for clamping (see "Extra pipe must be added at end..." and "Insufficient straight pipe between bends..." messages below.). If the pipe is bent, the elongation of the pipe during bending will be taken into account when reporting this error.

Pipe is too short

Pipe length is less than the company standard for pipes welded on both ends.

Spool is too long

The overall length of the spool exceeds company standards. The length is measured along the main axis of the spool.

Spool is too wide

The "width" in the spool is measured perpendicular to the main axis. This message indicates that the width exceeds company standards. Checks for surface treatment tank size are done separately.

Pipe Bending Messages**Bend angle too large**

A pipe bend exceeds the maximum bend angle that the bending machine can make.

Bend angle too small

A pipe bend angle is less than the company standard for bends. That is, the pipe is almost straight.

Bends have different radii, not allowed

Pipe has two or more bends, and they do not all have the same bend radius specified.

Bend radius too large or points too close together

The pipe geometry is physically impossible. This is an extreme case of the "insufficient straight pipe between bends..." problem. This problem may be fixed by moving a bend or using a smaller bend radius.

Extra pipe must be added at end, pipe end too short for bending machine to clamp onto

The bending machine needs a sufficiently long straight section at the start and end of the pipe to clamp onto (start), and support (end). This is a warning; bending can be done with a longer piece of pipe, with the extra pipe cut off afterwards.

Insufficient straight pipe between bends for bending machine to clamp onto

The bending machine needs a sufficiently long, straight section between bends to clamp the pipe. This problem may be fixed by moving a bend, by using a smaller bend radius, or by splitting the pipe and putting a joint between the bends.

No pipe bending machine for pipe diameter and bend radius

None of the pipe bending machines listed in the pipe shop XML data file have bend dies listed for the pipe diameter and bend radius of this pipe.

Pipe hits machine or floor during bending

The pipe cannot be bent because the free end or part of the pipe would hit the bending machine or the shop floor during bending. The simulator will have evaluated bending the pipe starting from either end before reporting this error, and will have checked all available bending machines with bend dies of the right size.

NOTE If the pipe has flanges on either end, and the **Flange Welding** check option has been selected, the simulator also checks for attached flange (if any) hitting the machine during bending.

Pipe and Spool Complexity**Too many bends in pipe**

The number of bends in the pipe exceeds the company standard. This limit is normally set to maintain dimensional accuracy standards because of accumulated error during bending.

Too many branches

The number of branches off of a main exceeds the company standard limit.

Too many planes in spool

The spool is difficult to fabricate because it is geometrically complex. That is, the number of independent planes exceeds the company standard for spools.

Treatment Tank**Spool too large for treatment tank**

A spool that requires galvanization or other tank treatment is too large for the tank.

No treatment tank found for <reatment_name> required by pipe spool

No treatment tank has been given in PipeBenders.xml for the treatment type called out by the pipe specification. This is an error in the editing of the PipeBenders.xml file, not a design error in the piping.

Individual pipes in spool require different tank treatments

A pipe spool is composed of multiple pipes that reference different pipe specs, and those pipe specs call out different, conflicting tank treatments.

Flange Welding**Flange hits machine or floor during bending, weld after bending**

Flanges cannot be welded on the pipe before bending because they would hit the bending machine or the shop floor during bending. The simulator will have evaluated bending the pipe starting from either end before reporting this error, and will have checked all available bending machines with bend dies of the right size.

Pipe too long for automatic flange welding

This is a warning that a straight pipe with flanges on one or both ends is too long to be put through the automatic flange welder, and must be manually welded.

Pipe too short for automatic flange welding

This is a warning that a straight pipe with flanges on one or both ends is too short to be put through the automatic flange welder, and must be manually welded.

Inside Grinding and Painting**Pipe too long for inside coating**

A pipe requires internal coating, but the pipe is too long for the available Pipe Internal Sprayer.

No internal sprayer found for inside coating

A pipe requires internal coating (as defined in the pipe spec entry in the XML file), but there is no Pipe Internal Sprayer available for the required coating material or for the pipe diameter.

Inaccessible for internal coating

A pipe that requires internal coating after bending has two or more bends. The sections between the bends are inaccessible for coating.

Bend on branch not allowed

A branch weld is inaccessible for grinding after welding because of a bend in the branch pipe.

Branch too far from end of main

Grinding cannot be done on a branch connection because it is too far from the end of the pipe.

Planning Inconsistencies Solution Messages

First Meet Check

First Meet assembly does not match the Planning Joint assembly

The planning joint is not located under the assembly containing the parts joined by the planning joint.

Hierarchy Check

Planning joint is not in an assembly that contains the two joined parts

The planning joint is located neither under the assembly containing the parts joined by the planning joint nor under a parent assembly/block of the assembly.

Physical Connection Check

Weld name is unknown

The weld bevel of a physical connection is not known because of a change to its associated planning joint.

Production Equipment Check

Production Equipment is not assigned to the assembly

The assembly has no production equipment assigned or the assigned production equipment is not compatible and cannot perform the weld.

Weld Side Check

The Weld Side of the Planning Joint is incorrect

The weld side of the planning joint is incorrect.

SECTION 19

Pre-Nesting

Pre-Nesting shapes do not use the physical connections to define the length of plates, profiles, and members. Instead, they use the connected part mounting angle to define the length. For profiles, the system allows you to control some corrections to the Pre-Nesting length of the profile.

Pre-Nesting is a plug-in service for Structural Manufacturing. You control the settings associated with Pre-Nesting by bulk loading the appropriate files.

After Pre-Nesting is enabled, you can create the following objects with the light (undetailed) plate and profile parts:

- Marking lines
- Shrinkage
- Margins
- Manufacturing plates
- Manufacturing profiles
- Service manager.

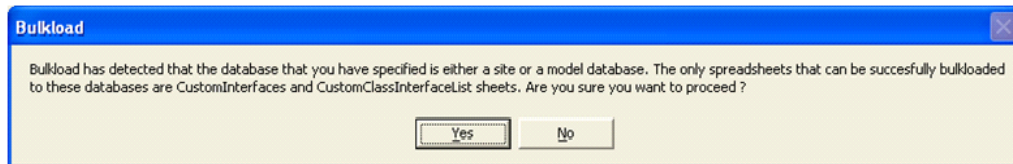
NOTE Manufacturing profiles use a plug-in rule to control the end-cut impact.

Pre-Nesting Setup

1. Bulk load the following Excel workbooks to the site and site schema databases in **Add, Modify and Delete** mode:

[Product
Folder]\ShipCatalogData\BulkLoad\DataFiles\StructManufacturing\Pre-Nesting\AllStructMfg
CodeLists.xls
[Product
Folder]\ShipCatalogData\BulkLoad\DataFiles\StructManufacturing\Pre-Nesting\StructMfgCu
stomClasses.xls

2. Click **Yes** when prompted with the following message box.



3. Select the appropriate **Prenesting Process** value on the **General** tab of the **Ship Properties** dialog box. The available options include:
 - **Pre-nesting (from Molded Forms definition)**
 - **Pre-nesting (from Detailing definition, if applicable)**
 - **Production nesting**

APPENDIX A

Structural Manufacturing Error Log Enable

Controls registry flags and registry values that enable or disable certain features in the Structural Manufacturing task.

This utility is delivered by default to the following location:

<Product Folder>\StructManufacturing\Client\Bin\ManufacturingErrorLogEnable.exe

When you close the utility, all changes are saved to the registry. The next time you start the utility, your new settings display.

See Also

Shrinkage Tab (Structural Manufacturing Error Log Enable Dialog Box) (on page 368)

Part Monitor Tab (Structural Manufacturing Error Log Enable Dialog Box) (on page 369)

Service Manager Tab (Structural Manufacturing Error Log Enable Dialog Box) (on page 370)

Unfolding Tab (Structural Manufacturing Error Log Enable Dialog Box) (on page 371)

Performance Tab (Structural Manufacturing Error Log Enable Dialog Box) (on page 371)

Geometry Tab (Structural Manufacturing Error Log Enable Dialog Box) (on page 374)

Shrinkage Tab (Structural Manufacturing Error Log Enable Dialog Box)

Enables or disables logging of data related to shrinkage rules.

Enable Shrinkage Rule logging

Turns on or off the logging of shrinkage rule information. By default this option is not selected.

Assembly Shrinkage Rule log file name

Indicates the location of the log file. By default, this is
[Drive]:\Temp\AssemblyShrinkageRule.log.

Part-Assembly based Shrinkage Rule log file name

Indicates the location of the log file. By default this is
[Drive]:\Temp\AssemblyPartShrinkageRule.log.

Part Monitor Tab (Structural Manufacturing Error Log Enable Dialog Box)

Controls various features of the Part Monitor. For more information, see *Part Monitor* (on page 272).

Part Monitor Attribute Viewer will take up <fraction> of the window

Indicates the ratio of Part Monitor to total screen. This is calculated from the Part Monitor's usual position on the right side of the window. The default value is **1/4**.

Activate Part Editor

Indicates whether the Part Editor is available. By default, this option is not selected.

Include GUIDs in reports

Indicates whether GUID data is included with the rest of the RAD data. The GUIDS always change which can cause confusion if you are using a comparison engine to look for very specific problems in the saved or loaded data. By default, this option is not selected.

Dump SHA file

This option is not currently available.

RAD Debug Reporting

Indicates whether the RAD data for RAD objects and attributes is dumped just before the RAD objects are converted back to Geom2D objects for persistence. This confirms that the attributes loaded are the same attributes as what was saved, and proves that persistence is working. You can use this to track down any problems. By default, this option is not selected.

Use System Temp

Indicates whether the data is forced to the System %Temp% folder on the client computer. This option is not currently available.

RAD Object Report path

Specifies the path that the software uses for writing the **RAD Debug Reporting** data. By default, this box is blank.

Detailed performance tracking

Enable All Performance Tracking

Indicates whether all of the performance tracking options are selected. By default, this option is not selected.

Master Load-time

Indicates whether over-all performance numbers are tracked. This option is required to enable any of the other **Detailed performance tracking** options. By default, this option is not selected.

Symbols

Indicates whether performance times for the placement of symbols, rotation, scaling, moving, and so on are tracked. By default, this option is not selected.

Supporting Functions

Indicates whether performance times for the supporting functions are tracked. Supporting functions are the required functions that do not fit into any of the other categories. For example:

- Finding points on objects, end-points, start-points, and center-points
- Getting system settings and selector rule options
- Setting up and activating layers
- Managing styles
- Grouping and ungrouping objects

By default, this option is not selected

Get XML Data

Indicates whether performance times are tracked for getting the XML data and processing it before it is loaded. By default, this option is not selected.

Annotations

Indicates whether performance times are tracked for the placement of annotations, rotation, scaling, moving, and so on. By default, this option is not selected.

RAD Functions

Indicates whether performance times are tracked for RAD functions. By default, this option is not selected.

Load-time Drawing

Indicates whether performance times are tracked for drawing data in the **Part Monitor**, **Annotation Editor**, or **Part Editor**. Examples include drawing lines, arcs, b-splines, circles, contours, markings, annotations, and so on. By default, this option is not selected.

Attributes

Indicates whether performance times are tracked for setting attributes on all RAD objects. This includes contours, markings, symbols, annotations, dimensions, and so on. By default, this option is not selected.

Service Manager Tab (Structural Manufacturing Error Log Enable Dialog Box)

Controls options for **Manufacturing Service Manager**. For more information, see *Manufacturing Service Manager* (on page 317).

Manufacturing Service Manager Trace

Indicates whether the **Manufacturing Service Manager** process log is generated. By default, this option is not selected.

Output file name

Specifies the log file name and location for the **Manufacturing Service Manager** trace. By default, this value is [Drive]:\Temp\MfgUpdateTrace.txt.

Unfolding Tab (Structural Manufacturing Error Log Enable Dialog Box)

Controls options for enabling or disabling the unfolding option.

Get Neutral Surface Topology using

Overrides the options selected in the *Manufacturing Plate* (on page 204) command. As part of survivability, Structural Manufacturing uses various surfaces for unfolding. By default, this value is set to **Detailing Neutral Surface**.

SNU

SNU unfold dump

Indicates whether the SNU unfold information is tracked. By default, this option is not selected.

Generate single surface

Indicates whether all the patches (surfaces) that are on the neutral surface are merged into a single b-spline surface and sent as an input to SNU. By default, this option is not selected.

Dump output folder

Specifies the root folder in which the software generates all of the curve and surface data passed to the SNU unfolding algorithm. The software creates a subfolder at this location with the plate name as the folder name. This data can be useful for the SNU developer to debug any problems in their algorithm. By default, this option is [Drive]:\Temp\SNUOutput.

GSTracker Options

MfgPlate/MfgProfile unfolding semantic log

Indicates whether to omit user-defined (custom) texts in the manufacturing plate or manufacturing profile unfolding semantic log files.

Performance Tab (Structural Manufacturing Error Log Enable Dialog Box)

Controls options for logging performance numbers for various functions and modules.

Tracks the time taken by critical functions while creating Manufacturing objects

Log to file

Indicates whether the log file is created. Performance numbers are not saved unless this option is selected. By default, this option is not selected.

Overwrite file

Indicates whether the log file is over-written each time the Structural Manufacturing Tracker is run and writes to the file. If not selected, the data is appended to the file. If selected, the file is

erased and new data is written to the file each time. By default, this option is not selected.

Print Summary of function calls

Indicates whether the summary of the function calls at the end of a given block of code is written to the log file. By default, this option is not selected.

The following example shows the summary in red:

```
+ Entering function Class_Initialize
  - call GetRuleValue took 0.01 secs
  - call GetRuleValue took 0.00 secs
  - call GetRuleValue took 0.00 secs
  - call GetRuleValue took 0.00 secs
  - call AddDialorBarTo2DEnv took 0.08 secs
  - call ResizeAttributeViewer took 0.03 secs
  - call ResizeAttributeViewer took 0.03 secs
  - call onToggleLayer took 0.00 secs
  - call onToggleLayer took 0.00 secs
  - call onToggleLayer took 0.02 secs
  - call onToggleLayer took 0.00 secs
  - call onToggleLayer took 0.01 secs
  - call onToggleLayer took 0.00 secs
  - call m_oAppEv_ActiveSheetChanged took 0.02 secs
  - call m_oAppEv_ActiveSheetChanged took 0.00 secs
  - call m_oAppEv_ActiveSheetChanged took 0.03 secs
  - call m_oAppEv_ActiveSheetChanged took 0.02 secs
  - call m_oAppEv_ActiveSheetChanged took 0.03 secs
  - call m_oAppEv_ActiveSheetChanged took 0.02 secs
  - call m_oAppEv_ActiveSheetChanged took 0.03 secs
  - call m_oAppEv_ActiveSheetChanged took 0.03 secs
  - call m_oAppEv_ActiveSheetChanged took 0.47 secs
  - call FileExists took 0.00 secs
  - call FileExists took 0.00 secs

+ Entering function Class_Initialize
  - call ResizeAttributeViewer took 0.02 secs
+ Exiting Function:Class_Initialize time: 0.58 secs

- call ShowHideGraphicViews took 0.08 secs
- call ShowHideGraphicViews took 0.08 secs
- call RestoreViewStates took 0.00 secs
- call RestoreViewStates took 0.00 secs
- call ShowHideGraphicViews took 0.00 secs
- call ShowHideGraphicViews took 0.00 secs
- call RestoreViewStates took 0.00 secs
- call RestoreViewStates took 0.00 secs
* Function GetRuleValue called 7 times, total time: 0.01 secs
* Function ResizeAttributeViewer called 2 times, total time: 0.84 secs
* Function RestoreViewStates called 2 times, total time: 0.00 secs
* Function Show3DUI called 2 times, total time: 0.00 secs
* Function ShowGraphicViews called 2 times, total time: 0.00 secs
* Function ShowHideGraphicViews called 2 times, total time: 0.08 secs
* Function m_oAppEv_ActiveSheetChanged called 12 times, total time: 0.69 secs
* Function onToggleLayer called 13 times, total time: 0.03 secs
+ Exiting Function:Class_Initialize time: 7.50 secs
```

Output File name

Specifies the file name and location for the performance log. By default, this value is C:\Temp\StrMfgTimer.log

Geometry Tab (Structural Manufacturing Error Log Enable Dialog Box)

Controls options for geometry.

Neutral Surface Topology Diagnostics Dump


Indicates whether the Structural Detailing **GetNeutralSurfaceTopology** function is called by the Structural Manufacturing task to get the neutral surface and contour information. If this option is selected, the software creates a subfolder in the %TEMP% path with the time stamp. This data can be useful to the Detailing team to analyze problems with their code. By default, this option is not selected.

Geometry Topology Trace Operations

Specifies how the geometry and topology functions handle logging. All inputs and outputs are written to a subfolder in the %TEMP% folder. This information is useful for the Manufacturing and Geometry and Topology teams to analyze any failures that occur. By default, this option is set to **No dump**.

APPENDIX B

Alma Act/Cut Nesting

Smart 3D provides export to and import from Alma act/cut nesting software. XML data for manufacturing plate parts, created with **Manufacturing Plate** , are exported to act/cut. In act/cut, the plate parts are nested on standard mill plate sizes, and plate remnants are maintained for future nesting. Finalized nesting XML data is then exported from act/cut and re-imported to Smart 3D. Smart 3D maintains the nesting status of the plates.

Alma act/cut (SKBY485AV-0306A or SKBY486AV-A306A) is a separately-purchasable third-party product available from Intergraph.

Glossary

abaft

Toward the stern of a ship, behind, further aft than.

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

access holes

An opening cut in the structure of a ship to permit entering or leaving various compartments.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

aft

Toward, at, or near the stern.

after body

The hull from aft of the midship section.

aftermost

Nearest the stern.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

auto-nesting

Automatic selection and location of various shapes and sizes of steel plate parts to fit on a standard steel plate, minimizing scrap while optimizing cutting time and minimizing thermal distortion.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bending

Process of conforming a plate to the hull surface. A bending process can be either simple (bent in one direction) or compound (bent in multiple directions). The software must minimize compound bending requirements.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

brace

A diagonal member used to stiffen a framework.

built ships

Complete database of NGC information after completion of the ship contract.

built-up member

A member built from multiple standard shapes to create a single, usually stronger, member.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

ceiling

Overhead design of the cabin area, including distribution systems for power, water, and ventilation.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

class rule check

Verification that the developing design meets the rules of a particular classification society, such as ABS, Lloyd's, or DNV.

Class Rules

Classification Society Design Rules.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

control point

A point that is used to control the shape of a NURBS curve or surface. Curves have a one-dimensional array of control points, while surfaces have a two-dimensional array.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cope

To cut out the top or bottom flanges and possibly the web so that one member frames into another.

cover plate

A plate used in building up flanges, in a built-up member, to give greater strength and area, or for protection.

cross section

The shape of a member when viewed along the member line.

cutback

An axial offset that typically represents the distance from a member centerline to its face. Cutbacks are used to account for the difference between how a structure is modeled and constructed in terms of lengths of members and quantities of materials.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

degree of freedom

An allowable direction of movement, either translation or rotation. There are six possible degrees of freedom (DOFs): translation X, Y, and Z, and rotation RX, RY, and RZ.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

external appendages

External structure attached to the hull, such as the propeller nozzle, shaft struts, bilge keel, and so forth.

fabricate

To cut, punch, and sub-assemble members in the shop.

fabrication plans

Structural detailed drawings of individual structural members, describing exactly how they are to be manufactured.

face

A topological object that represents a trimmed surface bounded by a loop of edges.

face plate

An edge reinforcement type that places a plate or profile at the selected plate edge.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fire integrity

Deck and bulkhead treatments and fire and smoke blocks for fire control and retardation.

flange

The projecting portion of a beam, channel, or column.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

jigs and fixtures

Design of, or data for, devices that position work or hold work in position for joining, transport, or erection.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joiner

Non-structural bulkheads, and trim and built-in furnishings.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

marking and cutting

Marking lines used to align any parts for assembly; also marks used to identify parts and cutting of standard plates into needed shapes, usually by N/C equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

member name

A user-definable alphanumeric code used to uniquely identify individual members in the model.

member part

A model object derived from the logical model that represents the manufactured physical member parts.

member system

A logical collection of member parts that can be moved as a single entity.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

natural surface

A surface without a boundary curve.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical occurrence

Unique specific object that has traceability and is the physical manifestation of an occurrence object. A physical occurrence applies to one and only one hull. It is a version of its occurrence object with as-built or as-modified differences included and has a serial number or lot number.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

product

Data objects that describe the components of a ship and any corresponding properties. An individual object or part (or its representation in the product model) that may be installed in the ship. Examples of individual products include objects such as a coffee urn, a light fixture, a piece of pipe, a piece of ventilation duct, a radar display console, a bulkhead plate, and a structural profile stiffening a bulkhead.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

sheetbody

A topological object that represents a collection of faces joined along their common edges (stitched).

shell structure

External portion of the surface of the plant.

ship

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Ship points to a Catalog (optionally shared with other Ships). Access control is managed at the Ship level.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specifications

Contracted requirements for the plant.

steel outfitting

Internal structural elements of a ship that are required to meet a local requirement such as foundations, non-structural bulkheads, walkways, and so forth.

stern frame

Casting and structure that support the rudder and shaft opening.

stiffener

An angle, plate, or channel fastened to a member to prevent buckling.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

swash bulkhead

A longitudinal or transverse nontight bulkhead in a tank that decreases the swashing motion of the liquid contents. A plate in a tank that has this same effect but that does not extend to the bottom of the tank is called a swash plate.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transverse

At right angles to the fore-and-aft center line.

transverse frames

The athwartship members that form the ribs of the ship.

trim

The difference between the forward draft and the aft draft.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

tumble home

The inboard slope of the side of a ship, usually above the designed waterline.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

vertical keel

A row of vertical plates extending along the center of the flat plate keel.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

water line

A line parallel with the base line that depicts the water.

watertight door

A door that when closed prevents the passage of water.

weather deck

A deck exposed to the weather.

weathertight door

A door that when closed prevents the passage of rain and spray.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

windlass

The machine used to hoist and lower anchors.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

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